Ellipse & Conic Tools

This chapter contains information on using the **Ellipse** tools and **Conic** tools. As you create an ellipse, the coordinate locations, lengths of the control rectangle and angles appear in the Status Line. As you create a conic, the rho value appears in the Status Line. These objects are also drawn with the current pen specifications for color, weight and pattern.

Referal: The rho value definition can be found on pages 11-6 and 11-7.

For each tool you can enter values in the Status Line to define an object, either before or after you create the object. If you enter the values after you select the tool but before you create the object, your first click in the drawing area automatically registers all Status Line values. If you enter values in the selected Status Line data field after creating the object and while the object is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the object to reflect the new values.

The tools explained here include:

- **2-Point Center Ellipse**
- **Opposite Corner Ellipse**
- **3-Point Center Ellipse**
- **3-Corner Ellipse**
- **2-Point Conic**
- **3-Point Conic**

- 4-Point Conic

Ellipse Tools

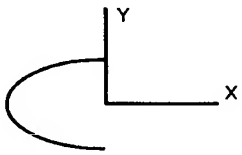
You can create ellipses with four tools: **2-Point Center**, **Opposite-Corner**, **3-Point Center** and **3-Corner**. These tools in the main tool palette construct ellipses inscribed within an invisible rectangle or parallelogram. The **2-Point Center Ellipse** tool uses the center point and one corner location of the rectangle. The **Opposite-Corner Ellipse** tool uses opposite corners of a rectangle. The **3-Point Center Ellipse** tool uses the center point, the midpoint of a side and the corner of the parallelogram. The **3-Corner Ellipse** tool uses three corners of a parallelogram.

Ellipses are defined by their *Major* and *Minor* Diameters, the Start and End Angles and their centers. The Start and End Angles specify the start and end location of the ellipse measured from the major axis.

Tip:

Vellum 3D Users: Ellipses are enhanced in your Designer Elements program and are now defined by centers.

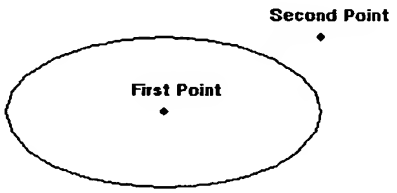
The graphic here shows an ellipse with a Start and End Angle of 90° and 180°.



The ellipse is drawn with the current pen specifications for color, weight and pattern.

2-Point Center Ellipse Tool

This tool constructs an ellipse inscribed within a rectangle defined by two specified points: the center point and one corner of the rectangle.



Using the 2-Point Center Ellipse Tool

1. Select the tool. The Message Line reads: *2-Point Center Ellipse: Pick center of ellipse. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*].*
2. Indicate the center of the ellipse.

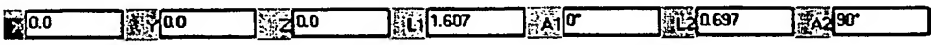
   Notice that the Message Line displays the next step for using the tool.

3. Indicate the corner of the rectangle defining the ellipse.

   If the two points are on the vertical or horizontal axis, a straight line is drawn.

You can create a copy of the last ellipse by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where you want the center. The copy is placed on the current work layer.

The Status Line allows you to specify the X, Y and Z coordinates of the center point and the length and angle of the semi-major and semi-minor axes of the ellipse.

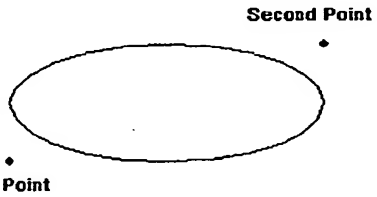


Geometric Characteristics

A 2-point center ellipse is created by placing the center point and corner of the control rectangle. It is made up of the following characteristics: Major Diameter, Minor Diameter, Start Angle, End Angle, and Center (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the ellipse and choose ***Window>Edit Objects*** or double-click on the ellipse.

Opposite-Corner Ellipse Tool

This tool draws an ellipse inscribed in a rectangle specified by opposite corners.



Using the Opposite-Corner Ellipse Tool

1.Select the tool. The Message Line reads: *Opposite-Corner Ellipse: Pick first corner of rectangle. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*].*

2.Indicate one corner of the rectangle defining the ellipse.

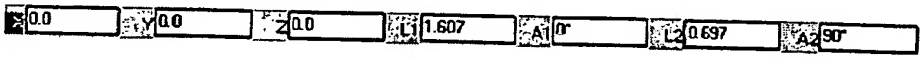
Notice that the Message Line displays the next step for using the tool.

3. Indicate the opposite corner of the defining rectangle.

   If the two points are on the vertical or horizontal axis, a straight line is drawn.

You can create a copy of the last ellipse by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking to place the lower-left point. The copy is placed on the current work layer.

The Status Line allows you to specify the X, Y and Z coordinates of the lower-left point, the length of the major and minor axes, L1 and L2 and angle of the major and minor axes, A1 and A2 of the ellipse.

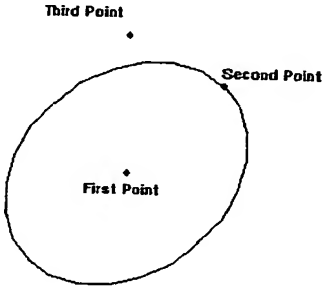


Geometric Characteristics

An opposite-corner ellipse is created by placing two points that represent the corners of the control rectangle and is made up of the following characteristics: Major Diameter, Minor Diameter, Start Angle, End Angle and Center (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the ellipse and choose ***Window>Edit Objects*** or double-click on the ellipse.

3-Point Center Ellipse Tool

This tool constructs an ellipse inscribed within a parallelogram calculated from three specified points: a center point, a midpoint of a side and a corner of the parallelogram.



Using the 3-Point Center Ellipse Tool

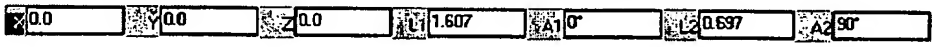
1. Select the tool. The Message Line reads: *3-Point Center Ellipse: Pick center of the ellipse. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*].* The Message Line will guide you through each successive step.

2. Pick the center of the ellipse.

3. Pick the midpoint of the side of the control parallelogram defining the ellipse.
4. Pick the corner of the control parallelogram defining the ellipse.

If the three points are on the vertical or horizontal axis, a straight line is drawn. You can create a copy of the last ellipse by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where you want the center. The copy is placed on the current work layer.

The Status Line allows you to specify the X, Y and Z coordinates of the center point and the length and angle of the sides of the parallelogram.

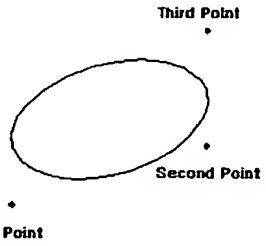


Geometric Characteristics

A 3-point center ellipse is created by placing the center point, midpoint of the side and the corner of the control parallelogram. It is made up of the following characteristics: Major Diameter, Minor Diameter, Start Angle, End Angle and Center (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the ellipse and choose ***Window>Edit Objects*** or double-click on the ellipse.

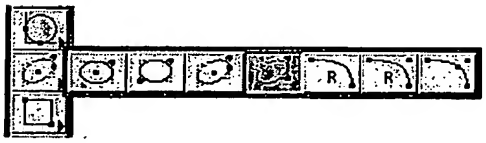
3-Corner Ellipse Tool

This tool draws an ellipse inscribed in a parallelogram defined by three corners.



Using the 3-Corner Ellipse Tool

1. Select the tool. The Message Line reads: *3-Corner Ellipse: Pick first corner of control parallelogram. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*. The Message Line will guide you through each successive step.

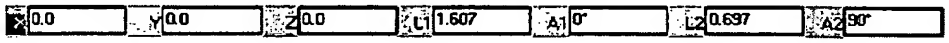


2. Pick one corner of the parallelogram defining the ellipse.
3. Pick another corner of the defining parallelogram.

4. Pick the final corner of the defining parallelogram.

If the three points are on the vertical or horizontal axis, a straight line is drawn. You can create a copy of the last ellipse by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking to place the lower-left corner of the parallelogram. The copy is placed on the current work layer.

The Status Line allows you to specify the X, Y and Z coordinates of a corner and the length and angle of the sides of the parallelogram.



Geometric Characteristics

A 3-corner ellipse is created by placing the three points that represent the corners of the control rectangle and is made up of the following characteristics: Major Diameter, Minor Diameter, Start Angle, End Angle and Center (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the ellipse and choose ***Window>Edit Objects*** or double-click on the ellipse.

Modifying Ellipses

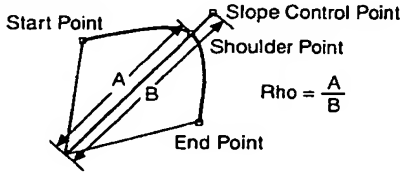
You can modify any ellipse in a number of ways:

- After it's created and while it's still selected by entering new values in the Status Line
- Using the **Transformation** tools (see Chapter 25)
- Using Edit Objects
- Select a control point and dragging it to a new location. The number of control points available to edit is based on the tool used to create the ellipse. A 2-Point Center ellipse has 2 control points.

Conic Tools

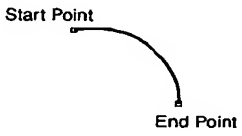
You can create conics with three tools: **2-Point Conic, 3-Point Conic** and **4-Point Conic**. The **2-Point Conic** tool uses the start point and end point to define the conic. The **3-Point Conic** tool uses the start point, end point and slope control point to define the conic. The **4-Point Conic** tool uses the start point, end points,

slope control point and shoulder point to define the conic. The *rho* value is the dimension ratio of the distance from the center point to the shoulder point and the center point and the slope control point. The graphic below illustrates the definitions of the conic characteristics.



When using a conic tool a rubberband image appears after you place all but the last point. This allows you to see the ellipse before it is drawn. The conic is drawn with the current pen specifications for color, weight and pattern.

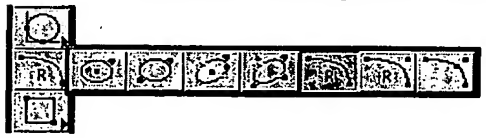
2-Point Conic Tool



This tool constructs a conic calculated from three specified points: the start point, end point and slope control point.

Using the 2-Point Conic Tool

1. Select the tool. The Message Line reads: *2-Point Conic: Pick start point. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*. The Message Line will guide you through each successive step.

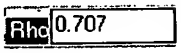


2. Pick the start point of the conic. As you move the cursor to place the second point, a rubberband image appears.
3. Pick the end point to complete the conic.

   You can modify the Rho value in the Status Line and press ENTER (Windows) or RETURN (Macintosh) to accept the new rho value. Rho values must be between .501 and .999. A rho value of .501 will create a straight line. A rho value of .999 will create a conic with a 90° angle.

You can create a copy of the last conic by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking to indicate the start point. The copy is placed on the current work layer.

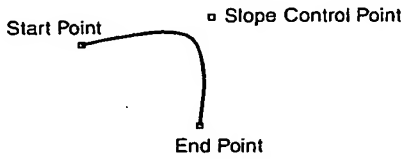
The Status Line allows you to specify the Rho value of the conic.



Geometric Characteristics

A 2-point conic is created by placing two points; the start point and end point and is made up of the following characteristics: Rho, End 1 (X, Y and Z values) and End (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the conic and choose ***Window>Edit Objects*** or double-click on the conic.

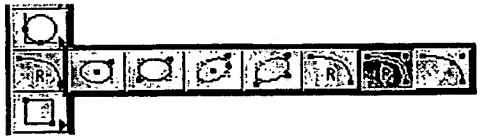
3-Point Conic Tool



This tool constructs a conic calculated from three specified points: the start point, end point and slope control point.

Using the 3-Point Conic Tool

1. Select the tool. The Message Line reads: *3-Point Conic: Pick start point. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*. The Message Line will guide you through each successive step.



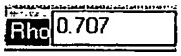
2. Pick the start point of the conic.
3. Pick the end point of the conic. Once you place this third point, a rubberband image appears.
4. Pick the shoulder point to complete the conic.

   You can modify the Rho value in the Status Line and press ENTER (Windows) or RETURN (Macintosh) to accept the new rho value. Rho values must be between .501 and .999. A rho value of .501 will create a straight line. A rho value of .999

will create a conic with a 90° angle.

You can create a copy of the last conic by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking to indicate the start point. The copy is placed on the current work layer.

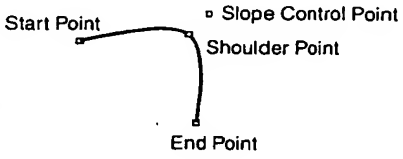
The Status Line allows you to specify the Rho value of the conic.



Geometric Characteristics

A 3-point conic is created by placing three points; the start point, end point and slope control point and is made up of the following characteristics: Rho, End 1 (X, Y and Z values) and End 2 (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the conic and choose ***Window>Edit Objects*** or double-click on the conic.

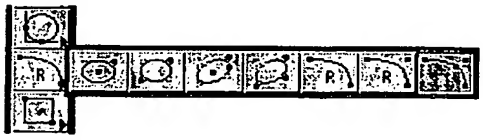
4-Point Conic Tool



This tool constructs a conic calculated from four specified points: the start point, end point, slope control point and shoulder point.

Using the 4-Point Conic Tool

1. Select the tool. The Message Line reads: *4-Point Conic: Pick start point. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*].* The Message Line will guide you through each successive step.

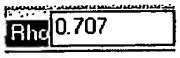


2. Pick the start point of the conic.
3. Pick the end point of the conic.
4. Pick the slope control point of the conic. Once you place this third point, a rubberband image appears.
5. Pick the shoulder point to complete the conic.

You can modify the rho value in the Status Line and press ENTER (Windows) or RETURN (Macintosh) to accept the new rho value. Rho values must be between .501 and .999. A rho value of .501 will create a straight line. A rho value of .999 will create a conic with a 90° angle.

You can create a copy of the last conic by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking to indicate the start point. The copy is placed on the current work layer.

The Status Line allows you to specify the Rho value of the conic.



Geometric Characteristics

A 4-point conic is created by placing four points; the start point, end point, slope control point and shoulder point and is made up of the following characteristics: Rho, End 1 (X, Y and Z values) and End 2 (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the conic and choose ***Window>Edit Objects*** or double-click on the conic.

Polygon Tools

These tools draw rectangles, inscribed polygons, circumscribed polygons, arbitrary polygons and polygons from curves. The polygon is drawn with the current pen specifications for color, weight and pattern.

For each tool you can enter values in the Status Line to define a polygon, either before or after you create the polygon. If you enter the values after you select the tool but before you create the polygon, your first click in the drawing area automatically registers all Status Line values. If you enter values in the selected Status Line data field after creating the polygon and while the polygon is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the polygon to reflect the new values.

Every polygon tool but the **Polygon from Curves** tool, provides two options for creating polygons, *Single Line* and *Smart Polygon*. These options are available through a pull-down menu in the Message Line.

Polygons created with the *Single Line* option, are composed of individual lines that can be modified independently. These polygons can be filleted/chamfered, extruded, revolved, offset or relimited.

Polygons created with the *Smart Polygon* option are true polygons, whose width and height can be modified. These objects cannot be filleted/chamfered, extruded, revolved, offset, or relimited with the 2D wireframe tools however you can perform 3D operations such as extrudes, blends and revolutions on these entities. To per-

form these operations on a smart polygon with 2D wireframe tools convert it into individual lines by choosing ***Edit>Change Object Type*** and select the line option. See Chapter 24, "Editing Commands," for more information.

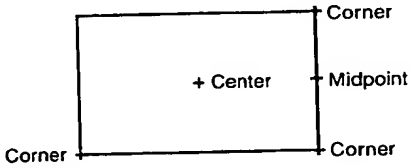
The tools explained here include:

- **Rectangle**
- **Inscribed Polygon**
- **Circumscribed Polygon**
- **Arbitrary Polygon**
- **Polygon from Curves**

Rectangle Tool

When you select the **Rectangle** tool, a subpalette appears in the Message Line containing four tools for creating rectangles, Polygon by center & point, Polygon by diagonals, Polygon by center & axis and Polygon by major & minor axes

The rectangle tools create rectangles from the selected points as directed by the specific tool.



Center/Corner Rectangle Tool

This tool draws a rectangle, using the center and corner you specify.

Using the Center/Corner Rectangle Tool

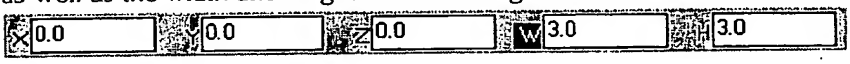
1. Select the **Rectangle** tool from the palette.

2. Select the **Center/Corner Rectangle** tool in the Message Line. The Message Line reads: *Rectangle: Pick center point of rectangle. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*].*
3. Pick the center point of the rectangle. As you move the cursor, a rubberband image appears.
4. Pick rectangle corner.

With the rectangle still selected, you can change the width and height of the rectangle by entering the values in the Status Line fields and pressing ENTER (Windows) or RETURN (Macintosh). You can also change the width and the height of a rectangle in the Edit Objects dialog box.

You can create a copy of the last rectangle by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where you want the upper-left corner.

The Status Line allows you to specify the X, Y and Z coordinates of the first point, as well as the width and height of the rectangle. Width is the active status field.



Diagonal Rectangle Tool

This tool draws a rectangle based on two corners along a diagonal.

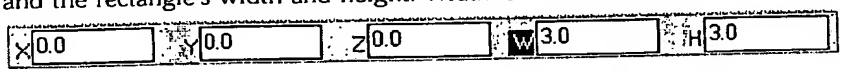
Using the Diagonal Rectangle Tool

1. Select the **Rectangle** tool from the palette.
2. Select the **Diagonal Rectangle** tool in the Message Line. The Message Line reads: *Rectangle: Pick first corner of rectangle. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*].*
3. Click one corner of the rectangle. As you move the cursor a rubberband image appears.
4. Click the opposite corner of the rectangle.

With the rectangle still selected, you can change the width and height of the rectangle by entering the values in the Status Line fields and pressing ENTER (Windows) or RETURN (Macintosh).You can also change the width and the height of a rectangle in the Edit Objects dialog box.

You can create a copy of the last rectangle by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where you want the upper-left corner.

The Status Line allows you to specify the X, Y and Z coordinates of the first point, and the rectangle's width and height. Width is the active field.



You can create a square by aligning the second point on the 45° construction line. If the two points are on the vertical or horizontal axis, a straight line is drawn. If you want to draw a square from the center rather than opposite corners, use one of the other polygon tools, specifying four sides. You can't create a rectangle from the center.

Center/Axis Rectangle Tool

This tool draws a rectangle using the center, midpoint and corner.

Using the Center/Axis Rectangle Tool

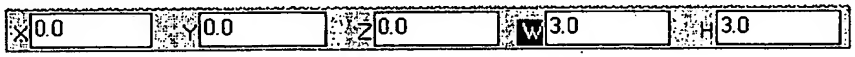
1. Select the **Rectangle** tool from the palette.
2. Select the **Center/Axis** tool in the Message Line. The Message Line reads: *Rectangle: Pick center point of rectangle. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*].*
3. Click the center point of the rectangle.
4. Click the midpoint of the rectangle side. As you move the cursor a rubberband image appears.
5. Click the corner of the rectangle.

With the rectangle still selected, you can change the width and height of the rectangle by entering the values in the Status Line fields and pressing ENTER (Windows) or RETURN (Macintosh).

You can also change the width and the height of a rectangle in the Edit Objects dialog box.

You can create a copy of the last rectangle by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where you want the upper-left corner.

The Status Line allows you to specify the X, Y and Z coordinates of the first point, as well as the width and height of the rectangle. Width is the active status field.



Major/Minor Axis Rectangle Tool

This tool draws a rectangle using the corners you specify.

Using the Major/Minor Rectangle Tool

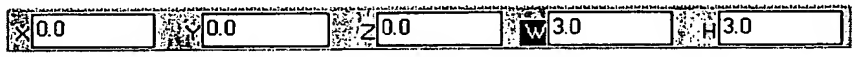
1. Select the **Rectangle** tool from the palette.
2. Select the **Major/Minor Axis** tool in the Message Line. The Message Line reads: *Rectangle: Pick center point of rectangle. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*].*
3. Click the first corner of the rectangle.
4. Click the second corner of the rectangle. As you move the cursor, a rubberband image appears.
5. Click the third corner of the rectangle.

With the rectangle still selected, you can change the width and height of the rectangle by entering the values in the Status Line fields and pressing ENTER (Windows) or RETURN (Macintosh).

You can also change the width and the height of a rectangle in the in Edit Objects dialog box.

You can create a copy of the last rectangle by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where you want the upper-left corner.

The Status Line allows you to specify the X, Y and Z coordinates of the first point, as well as the width and height of the rectangle. Width is the active status field.



Geometric Characteristics

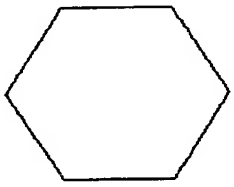
A rectangle polygon is created by picking the appropriate points as directed by the selected tool in the rectangle subpalette. A rectangle is made up of the following

characteristics according to the Edit Objects dialog box: Length and Width and includes the option to Show Frame and Show Fill.

Show Frame refers to the display of the polygon. Show Fill refers to the placement of a fill within the polygon. Setting Show Frame to No, turns off the wireframe display. (If the wireframe is not displayed and does not contain a fill, your geometry will become invisible when your click Apply.) Setting Show Fill to Yes, places a fill within the polygon, based on the default pattern and fill currently selected in the Pen menu. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the polygon and choose ***Window>Edit Objects*** or double-click on the polygon.

Inscribed Polygon Tool

This tool creates a polygon where the radius of the circumscribing circle determines the location of the polygon's vertices.



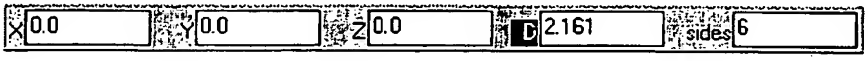
The default polygon is a hexagon, but you can specify the number of sides in the Status Line. The Status Line shows a diameter for the circle, the standard way of describing a polygon inscribed in a circle.

Using the Inscribed Polygon Tool

1. Select the tool. The Message Line reads: *Inscribed Polygon: Pick center of polygon. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*.
2. Click the center of the polygon. The Message Line reads: *Inscribed Polygon: Pick vertex of polygon.* As you move your cursor a rubberband polygon appears guiding your construction.
3. Pick a point on the circumference of the circumscribing circle to complete the polygon.

You can create a copy of the last inscribed polygon by holding down the CTRL (Windows) or OPTION (Macintosh) key and clicking where you want the center.

The Status Line allows you to specify the X, Y and Z coordinates of the center, the diameter of the circle defining the polygon, and the number of sides. Diameter is the active Status Line selection, and the default number of sides is six.



Geometric Characteristics

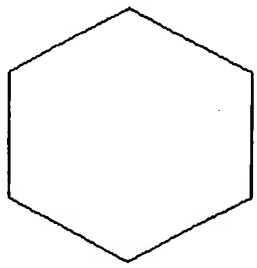
An inscribed polygon is created by picking the center and the vertex of the polygon. It is made up of the following characteristics according to the Edit Objects dialog box: Diameter, Sides, Scribe Type and includes the option to Show Frame and Show Fill.

Scribe Type refers to circle type used to define the polygon. Choosing Circle Outside places the polygon within the circle, defining an inscribed polygon. Choosing Circle Inside places the polygon on the outside of the circle, defining a circumscribed polygon.

Show Frame refers to the display of the polygon. Show Fill refers to the placement of a fill within the polygon. Setting Show Frame to No, turns off the wireframe display. (If the wireframe is not displayed and does not contain a fill, your geometry will become invisible when your click Apply.) Setting Show Fill to Yes, places a fill within the polygon, based on the default pattern and fill currently selected in the Pen menu. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the polygon and choose ***Window>Edit Objects*** or double-click on the polygon.

Circumscribed Polygon Tool

This tool draws a polygon in which the radius/diameter of the circle determines the midpoint of the sides. The default shape is a hexagon but you can specify the number of sides in the Status Line. Notice that the Status Line shows a diameter for the circle, the standard way of describing a polygon circumscribed around a circle.



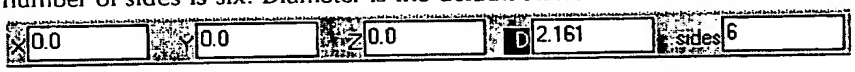
Using the Circumscribed Polygon Tool

1. Select the tool. The Message Line reads: *Circumscribed Polygon: Pick center of polygon. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*.
2. Click the center of the polygon and the midpoint of one of the sides (as directed by the Message Line).

   After you place the first point, a rubberband polygon appears guiding construction.

You can create a copy of the last circumscribed polygon by holding down the CTRL (Windows) or OPTION (Macintosh) key and clicking where you want the center.

The Status Line allows you to specify the X, Y and Z coordinates of the center, the diameter of the circle defining the polygon, and the number of sides. The default number of sides is six. Diameter is the default Status Line selection.



Geometric Characteristics

A circumscribed polygon is created by picking the center and the midpoint of the polygon side. It is made up of the following characteristics according to the Edit Objects dialog box: Diameter, Sides, Scribe Type and includes the option to Show Fill and Show Frame.

Scribe Type refers to circle type used to define the polygon. Choosing Circle Outside places the polygon within the circle, defining an inscribed polygon. Choosing Circle Inside places the polygon on the outside of the circle, defining a circumscribed polygon.

Show Frame refers to the display of the polygon. Show Fill refers to the placement of a fill within the polygon. Setting Show Frame to No, turns off the wireframe display. (If the wireframe is not displayed and does not contain a fill, your geometry will become invisible when your click Apply.) Setting Show Fill to Yes, places a fill within the polygon, based on the default pattern and fill currently selected in the Pen menu. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the line and choose ***Window>Edit Objects*** or double-click on the polygon.

Arbitrary Polygon Tool

This tool draws a polygon with the number of sides determined by the points you choose.

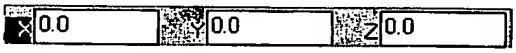
Using the Arbitrary Polygon Tool

1. Select the tool. The Message Line reads: *Arbitrary Polygon: Pick first point. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*].*
2. Click the next point (as directed by the Message Line).
3. Continue clicking points to complete the desired polygon. Single click the last point to close the polygon at its starting point. (If you try to double-click at this point you will receive an error message.)

   You can also double-click to place the last point (that is not the starting point of the polygon and your Designer Elements program will close the polygon automatically.

You can create a copy of the polygon by holding down the CTRL (Windows) or OPTION (Macintosh) key and clicking where you want the center.

The Status Line allows you to specify the X, Y and Z coordinates of the first point. X is the default Status Line selection.



Geometric Characteristics

An arbitrary polygon is created by placing the desired points for the polygon. It is made up of the following characteristics according to the Edit Objects dialog box: Defining Point and the options to Show Fill and Show Frame.

Defining Point refers to the chosen's point X, Y and Z location. The section below Defining Points lists the points with the X, Y and Z location. The selected point is displayed in the Defining Point fields. Each point can be edited individually.

Show Frame refers to the display of the polygon. Show Fill refers to the placement of a fill within the polygon. Setting Show Frame to No, turns off the wireframe display. (If the wireframe is not displayed and does not contain a fill, your geometry will become invisible when your click Apply.) Setting Show Fill to Yes, places a fill

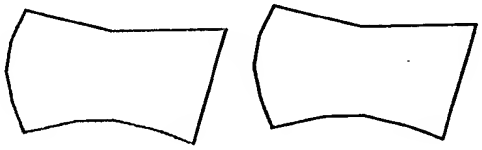
within the polygon, based on the default pattern and fill currently selected in the Pen menu. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the polygon and choose ***Window>Edit Objects*** or double-click on the polygon.

Polygon from Curves Tool

This tool creates a polygon from the curves you choose. The curves do not have to be connected to create a polygon. If these curves are not connected, the tool adds curves to complete the polygon. The polygon created when a curve is missing depends on the order of the curve selection.

Once the polygon is created a parent/child relationship has been established between the original curves and the polygon. The original curves selected are still available for editing and any change made will affect the polygon. Due to this same parent/child relationship, in order to move the polygon and retain the relationship, you must select the curves and the polygon. You can break the relationship by selecting them and choosing ***Edit>Remove Links***. See Chapter 24, "Editing Commands" for more information.

The graphic below on the left represents five different curves. The graphic on the right is the one polygon created after using the tool.



Using the Polygon from Curves Tool

1. Select the tool. The Message Line reads: *Polygon from Curves: Select a collection of curves [Shift = Extend].*
2. Select all the desired curves. If you are selecting the curves individually, hold down the SHIFT key before beginning your selection.

   A polygon is created from those curves.

There are no entries in the Status Line.

Geometric Characteristics

Using this tool a polygon is created by selecting specific curves. No new geometry is created. The following characteristics are listed in the Edit Objects dialog box: Show Fill and Show Frame.

Show Frame refers to the display of the polygon. Show Fill refers to the placement of a fill within the polygon. Setting Show Frame to No, turns off the wireframe display. (If the wireframe is not displayed and does not contain a fill, your geometry will become invisible when your click Apply.) Setting Show Fill to Yes, places a fill within the polygon, based on the default pattern and fill currently selected in the Pen menu. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the polygon and choose ***Window>Edit Objects*** or double-click on the polygon.

Spline Tools

The spline tools in the main tool palette create *NURBS* (Non-Uniform Rational B-Splines). These NURB splines are smooth curves created by a complex mathematical formula.

NURB splines provide designers with two interrelated functions. First, curvature continuity remains intact when the curve is changed. Kinks don't develop as the spline is altered. Second, NURB splines provide localized control of a complex curve by changing the control points.

Tech Note:

Your Designer Elements program creates B-Splines of the third and fourth order.

These properties are essential in aerodynamic designs. Air molecules moving over a wing surface must flow smoothly for maximum aero-dynamic lift. If the surface does not maintain curvature continuity, the air molecules separate from the wing surface and cause a vacuum. Such a vacuum causes an eddy as the molecules try to fill it. This disruption of air flow increases the drag, which is not a part of an effective design.

Complete curvature continuity also improves styling. The appearance of a car is one of the major sales factors. The potential buyer would not be impressed if the showroom lights' reflection on the car rippled and wavered. It is complete curvature continuity that makes a smooth reflection.

Localized control of complex curves allows you to make minor modifications without adversely affecting the shape. For example, if a new, bigger engine wouldn't fit under a perfectly-designed hood, you could use a NURB spline to raise the center of the hood without changing the basic design. NURB splines are also valuable for

injection mold designs to eliminate the swirl of plastic as it is injected into the mold. Such designs provide better surface finishes and allow thinner cross-sections in the die.

The spline creation tools are the **Through-Points B-Spline** tool, **Vector Spline** tool, **Bezier Spline** tool and Sketch Spline tool. At the most basic level these splines are the same. For the Through-Points B-Spline and the Bezier Spline, your Designer Elements program has the additional step of interpolating the points to calculate a control polygon which is hidden from you. For the Vector Spline the interpolation step is eliminated because you define the control polygon to which you have access.

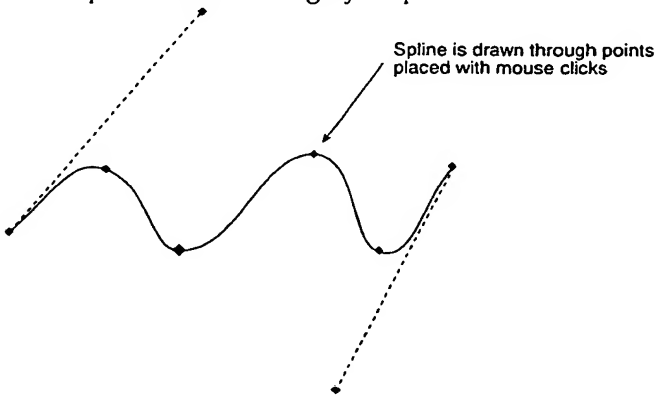
Spline control points automatically lock after placement. You can select one point and move it without affecting the other control points.

You can enter values in the Status Line to define a spline either before or after you create the spline. If you enter the values after you select the tool but before you create the spline, your first click in the drawing area automatically registers all Status Line values. If you enter values in the selected Status Line data field after creating the spline and while the spline is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the spline to reflect the new values. The tools explained here include:

- **Through-Points B-Spline**
- **Vector Spline**
- **Bezier Spline**
- **Sketch Spline**
- **Helix Curve**
- **Add Control Point**
- **Remove Control Point**
- **Modify Slope**
- **Elevate Curve**
- **Fair Curve Spline**

Through-Points B-Spline Tool

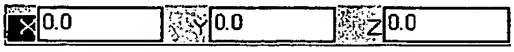
The **Through-Points B-Spline** tool draws a spline through your specified points. Use this tool when you want the B-Spline to interpolate a collection of points. The interpolation algorithm is based on predefined B-Spline blending functions, control point locations and imposing curvature continuity across the curve length. This results in a smooth spline created through your points.



Using the Through-Points B-Spline Tool

1. Select the tool. The Message Line reads: *Through-Points B-Spline: Pick control points. (End = ESC, Double-click).*
2. Click the points for the vectors of the spline.
3. Double-click the last point or hit the ESC key.

The Status Line shows the X, Y and Z coordinates for each point as it is placed.



If you create a spline that crosses over itself, the Drafting Assistant will not be able to find that intersection. This is by design.

Geometric Characteristics

A Through-Points B-Spline is created by picking the desired points. A spline is made up of the following characteristics according to the Edit Objects dialog box: Start and End Angles (relative to work plane) and Defining Point.

Angles refer to the degrees of the tangent relative to the work plane for the spline. You can change the Start and End Angles by clicking within the check box and entering the angle.

Defining Point refers to the X, Y and Z location of the active point. X, Y and Z values for each point can be edited individually using these fields.

This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the spline and choose ***Window>Edit Objects*** or double-click on the spline.

Vector Spline Tool

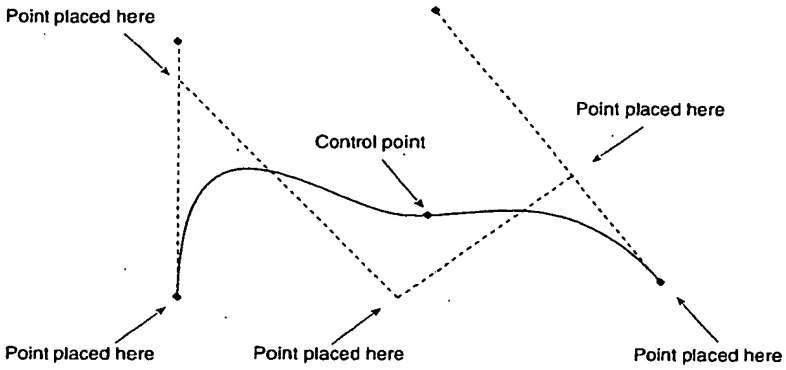
Tip:

:With the **Vector Spline** tool all original points used to create the spline will display when the spline is selected.

This tool draws a spline using vectors determined by the points you specify. The **Vector Spline** tool uses each point that you place as the vertex of a vector for the spline it creates. Vector splines are smooth curvature, continuous B-Splines created from a control polygon that you define. The polygon influences the shape (position and tangencies) of the spline. Except for the start and end points, the control polygon does not necessarily lie on the spline.

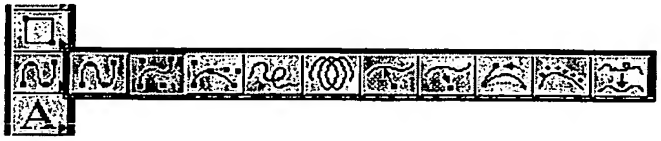
Choosing ***Edit>Show Points*** will display all the control points used to define the spline and can be selected to edit the spline. This provides the additional benefit of being able to smooth your spline to eliminate inflections. To determine whether or

not your spline has inflections, choose ***Verify>Curvature***. An inflection is located where the curvature changes from one side of the spline to the other.



Using the Vector Spline Tool

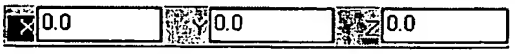
1. Select the tool. The Message Line reads: *Vector Spline: Pick control points (End = ESC, Double-click).*



2. Click the points for the vectors of the spline.
3. Double-click the last point.

This Designer Elements program uses these vectors to calculate the control points (two fewer than the number of vertices you specify). The spline is tangent to the first and last vectors and passes through the calculated control points.

The Status Line shows the X, Y and Z coordinates of each point as they are placed.

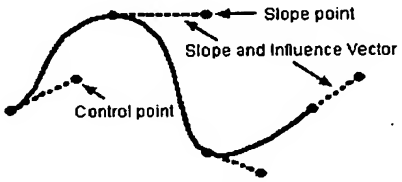


Geometric Characteristics

A vector spline is created by picking control points and is made up of the one characteristic according to the Edit Objects dialog box: the Defining Point (X, Y and Z location). Defining Point refers to the active point's X, Y and Z location. Each point can be edited using the X, Y, Z fields. This information is listed in the Edit

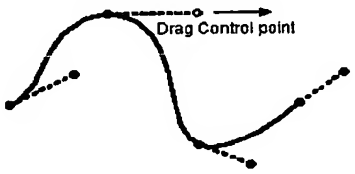
Objects dialog box under the Geometry tab. To display the dialog box, select the spline and choose ***Window>Edit Objects*** or double-click on the spline.

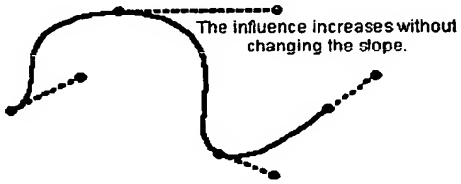
Bezier Spline Tool



This tool creates curvature continuous (C2) splines through a collection of user defined points. At each control point, you can control the direction and the magnitude of the tangent. After creating a spline with this tool, you can move the control points, change the slope (direction) and the influence of a control point using the spline vector.

The influence of a control point is determined by the length of the control point's vector. The longer the vector, the greater the influence a control point has on the spline. You can adjust the length of control point vector but maintain the slope. The left graphic here shows a Bezier spline, a selected control point and the direction it will be moved to change the influence but maintain the slope. The right graphic shows the spline with a longer vector.



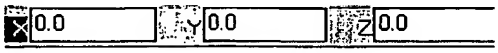


You can also use the **Add Control Point**, **Remove Control Point** and **Modify Slope** tools to change the spline.

Using the Bezier Spline Tool

1. Select the tool. The Message Line reads: *Bezier Spline: Pick control points (End = ESC, Double-click).*
2. Click the points of the spline.
3. Double-click the last point.

The Status Line shows the X, Y and Z coordinates of each point as they are placed.



Geometric Characteristics

A Bezier spline is created by picking control points and is made up of the following characteristics according to the Edit Objects dialog box: the Defining Point (X, Y and Z location) and the points list. This list includes the spline control points and the slope points (if displayed). Display the slope points by selecting the spline and choosing ***Edit>Show Points***. Defining Point refers to the active point's X, Y and Z location. Each point can be edited using the X, Y, Z fields. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the spline and choose ***Window>Edit Objects*** or double-click on the spline.

Sketch Spline Tool

The **Sketch Spline** tool samples points as the cursor moves and creates a smooth spline throught the sampled data. Once selected this tool has two options located in the Message Line, **Sketch Spline** tool and **Sketch on a Surface** tool.

Using the Sketch Spline Tool

1. Select the tool. The Message Line reads: Sketch Spline: Press and drag mouse (Option=AutoClose).
2. Click and drag the mouse on the screen.
3. Release the mouse button to create the sketch spline.

Using the Sketch on a Surface Tool

1. Select the tool. The Message Line reads: Sketch on Surface: Select surface or solid to sketch on (Option=AutoClose).
2. Select the surface or solid to sketch on.
3. Click and drag the mouse over the selected surface, keeping the points on the surface.
4. Release the mouse button to complete the sketch.

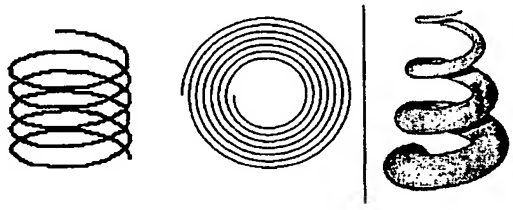
Tech Note:

If you want to sweep a surface on a helix, the helix must be created using this tool. See Chapter 16 for information on using the **Sweep Surface** tool.

Helix Curve Tool

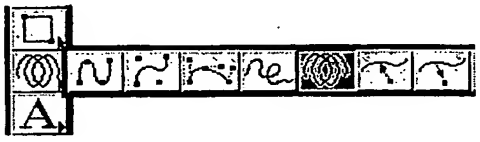
The **Helix Curve** tool draws a curve based on its start point, end point, *Pitch*, diameter, length and draft angle. The endpoint defines the length and orientation of the helix. Pitch is the distance between helix peaks (two adjacent turns of the helix). The number of turns in the helix is equal to the length of the helix divided by the pitch. The draft angle must be a value less than 90°.

You can create a standard helix (left graphic below), a spiral (middle graphic below) by entering zero for the helix length, or a swept solid using a helix with a draft as the sweep path (right graphic below).



Using the Helix Curve Tool

1. Select the **Helix Curve** tool. The Message Line reads: *Helix Curve: Pick start and end of helix [Ctrl* (Windows) or *Option* (Macintosh) = *Left handed helix].*



2. The Status Line contains the Pitch, Diameter, Length and Draft Angle data fields.

Enter the desired values in the appropriate fields. Tab between fields.

3. Pick the start and end point of the helix. A right handed helix is created.

While the helix is still selected you can change the Status Line values. Type the new values and press ENTER (Windows) or RETURN (Macintosh) and the helix updates.

Geometric Characteristics

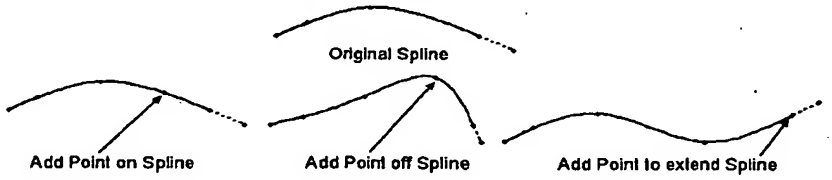
A helix is created by picking the start and end point and specifying the pitch and diameter. A helix is made up of the following characteristics according to the Edit Objects dialog box: the Diameter, Pitch, Length and Draft Angle.

You also have a check box to change the helix direction. By default, using this tool creates a right-handed helix. You can change the helix to a left-handed helix by deselecting the *Right handed helix* option and clicking Apply or Close. If you already created a left-handed helix, the box is not checked.

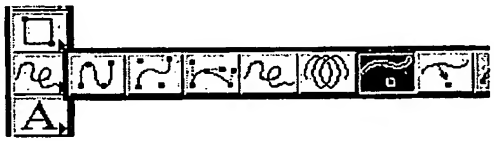
This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the helix and choose ***Window>Edit Objects*** or double-click on the helix.

Add Control Point Tool

This tool allows you to add another control point to an existing spline both within the spline and outside of the spline to extend it.



Using the Add Spline Control Point Tool



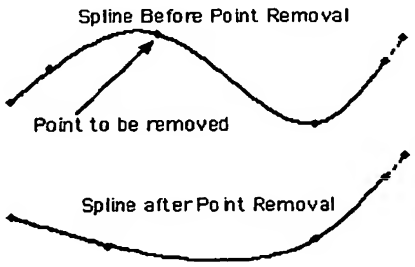
1. Select the tool. The Message Line reads: *Add Control Point: Select the spline to add point to (Shift = Extend).*
2. Click on the spline to which you want to add a point.
3. Pick the desired location(s) for the new control point(s).

Note: To see the new control point(s), as well as the existing control points, you must first select the spline and then choose ***Edit>Show Points*** (or use the Edit Objects dialog box to specify whether the Control points are visible or hidden).

There are no Status Line entries for this tool.

Remove Control Point Tool

This tool allows you to remove a control point from an existing spline. A spline must have more than two control points to use this tool.



Using the Remove Control Point Tool

1. Select the tool. The Message Line reads: *Remove Control Point: Select the spline point to remove (Shift = Extend).*
2. If the control points of the spline are not displayed, select the spline and choose ***Edit>Show Points.***
3. Pick the control point you want to remove.

There are no Status Line entries for this tool.

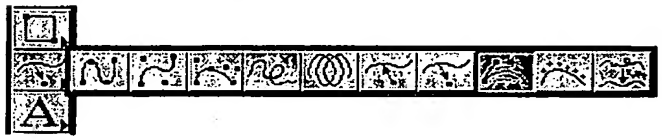
Modify Slope Tool

This tool allows you change the endpoint slope of an existing spline created using the **Through-Points B-Spline** tool. You have four options for changing the slope: *Free, Reference, Reverse* and *Explicit.*

Free	Frees the spline slope and returns it the original slope as defined by this Designer Elements program for the chosen slope control point. This is helpful if you've adjusted the slope of a spline in numerous locations along the spline and want to return to the spline that was originally created.
Reference	Uses the slope of other curves or surfaces as a reference to make the selected curves tangent.
Reverse	Reverses a curve 180°.
Explicit	Allows you to set the exact slope of the curve relative to the work plane (between 0° and 360°). A slope of 0° will make the curve tangent to the plane. Choose ***Planes>Show Work Plane*** when using this option to assist you.

Using the Modify Slope Tool - Free Option

1. Select the tool. The Message Line reads: *Modify Slope: Select spline slope(s) to free (Shift = Extend).*



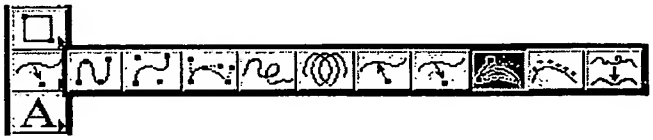
2. Choose the *Free* option in the pull-down menu of the Message Line.
3. If the control points of the spline are not displayed, select the spline and choose ***Edit>Show Points.***
4. Pick the spline slope control endpoint you want to free.

   The slope control endpoint returns to its original position.

There are no Status Line entries for this tool.

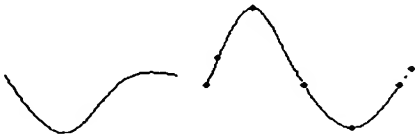
Using the Modify Slope Tool - Reference Option

1. Select the tool. The Message Line reads: *Modify Slope: Select the spline end to set slope at (Shift = Extend).*

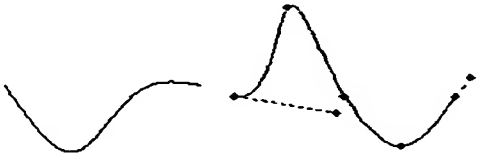


2. Choose the *Reference* option in the pull-down menu of the Message Line.
3. If the control points of the spline that you want modified are not displayed, select the spline and choose ***Edit>Show Points***.
4. Pick the spline want to modify. In the graphic here, it's the spline displaying points.
5. Pick the curve, surface or solids whose slope you want to reference.

   The icon now becomes a target icon.



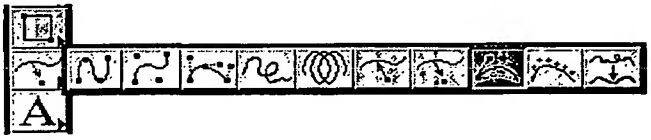
6. Click on the endpoint of the curve you want to modify, (The curve and the reference object do not have to connect.) The curve slope adjusts.



There are no Status Line entries for this tool.

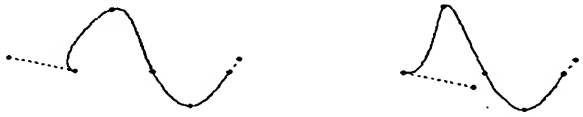
Using the Remove Control Point Tool - Reverse Option

1. Select the tool. The Message Line reads: *Modify Slope: Select spline slope point to reverse (Shift = Extend).*



2. Choose the *Reverse* option in the pull-down menu of the Message Line.

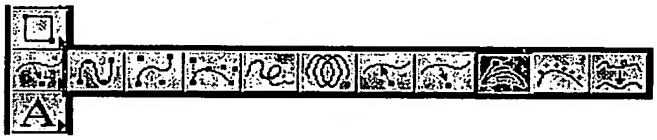
3. If the control points of the spline are not displayed, select the spline and choose ***Edit>Show Points***.
4. Pick the spline slope control endpoint you want to reverse, as in the left graphic below. The slope control endpoint reverses 180° (right graphic).



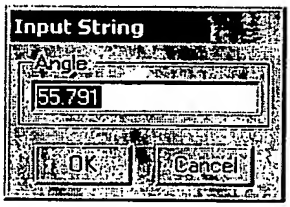
There are no Status Line entries for this tool.

Using the Modify Slope Tool - Explicit Option

1. Select the tool. The Message Line reads: *Modify Slope: Select spline slope point for explicit set (Shift = Extend).*



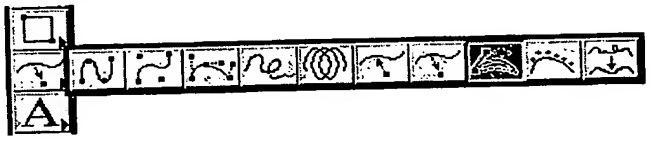
2. Choose the *Explicit* option in the pull-down menu of the Message Line.
3. If the control points of the spline are not displayed, select the spline and choose ***Edit>Show Points***.
4. Pick the spline control endpoint you want to set. An Input String dialog box appears.
5. Enter the Angle for the slope in the data field and click OK. The dialog box closes and the slope of the spline is adjusted. (Click Cancel if you want to close the dialog box without changing the angle value.)



There are no Status Line entries for this tool.

Using the Modify Slope Tool - Free Magnitudes Option

1. Select the tool. Choose the Free Magnitudes option in the pull down menu. The Message reads: Modify Slope: Select Spline Magnitude to free (Shift=*Extend*).



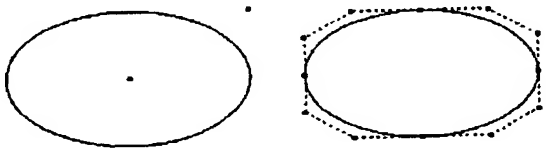
2. If the control points of the spline are not diplayed, select the spline and choose **Edit>Show Points**.
3. Select the bezier curve control point you want to free.

The Bezier slope resets to an optimally calculated slope.

Elevate Curve Tool

This tool elevates the degree of a curve, introducing more control points. These control points can be used to modify the curve.

Curves are defined by NURB polynomial equations. The most basic question possible is used to define the curve. Shapes defined by higher degree polynomials have more control points than those defined by lesser degree polynomials. This tool raises the degree level for a shape, providing more control points for curve manipulation. Curves can be elevated up to the 22nd degree. However, it is recommended that you never elevate a curve highter than 9 degrees. The left side of the graphic below shows the original ellipse. The right side of the graphic below shows the ellipse elevated one degree.



Using the Elevate Curve Tool

1. Select the Elevate Curve tool. The Message Line reads: Elevate Curve: Pick curve to elevate [Shift=Extend].
2. Select the curve to elevate.

The curve elevates one degree.

There are no entries in the Status Line.

Fair Spline Tool

The **Fair Spline** command provides a tool to globally smooth a curve. Fairing a spline optimally moves control vertices to locations that minimize large curvature variations. Curves eligible for fairing are limited to vector splines. It's helpful to watch the control vertices move as you fair the spline with the **Edit>Show Points** command. You can also use the **Verify>Curvature** tool to visually inspect the curvature graph of a curve.

Using the Fair Spline Tool

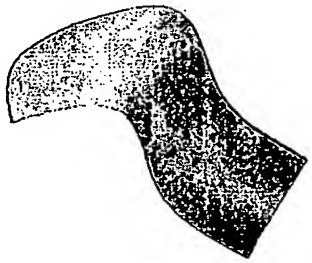
1. Select the tool. The Message Line reads: *Fair Spline: Select curve to fair.*
2. Click on the spline which you want to fair.

The status line displays a move tolerance value for the tool. The move tolerance is the maximum distance a spline control vertex will be allowed to move towards its optimal position.

Introduction to Surface Modeling

A surface/mesh model is a more complete and less ambiguous representation than a wireframe model. Surface models take the representation of an object one step beyond wireframe models by defining the area between selected boundaries with smooth equations. Mesh models also go beyond wireframe models but are defined by nodes or vertices. Consequently, surface and mesh modeling provides a means for the designer to create complex shapes such as cars, ships and aircraft.

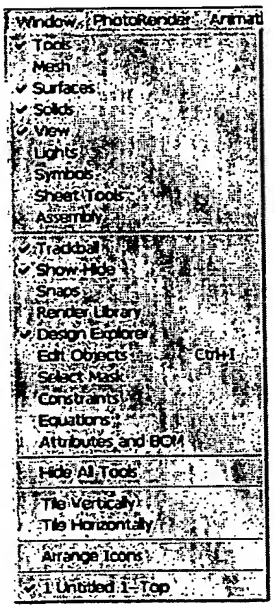
Your Designer Elements program surfaces are based on Spatial Technology's ACIS Geometry Engine. ACIS surfaces are composed of Non Uniform Rational B-Splines (NURBs). NURBs provide a highly precise yet flexible mathematical definition for modeling even the most demanding free form shapes. Surfaces generated with your Designer Elements program are suitable for precise geometric analysis and can be passed to computer aided manufacturing applications that support SAT or IGES file formats.



The topics discussed in this chapter include:

- **Surface Modeling** tools
- Surfaces, Selection and Display
- Surfaces and Instances
- Cutting and Pasting Surfaces
- Exporting Surfaces
- Object Types and Edit Objects

Surface Modeling Tools



The surface modeling tools are accessible from the **Surfaces** tool palette. The **Surfaces** tool palette automatically displays when you first launch your Designer Elements Program. If you close the palette and need to display the palette again, choose ***Window>Surfaces***.

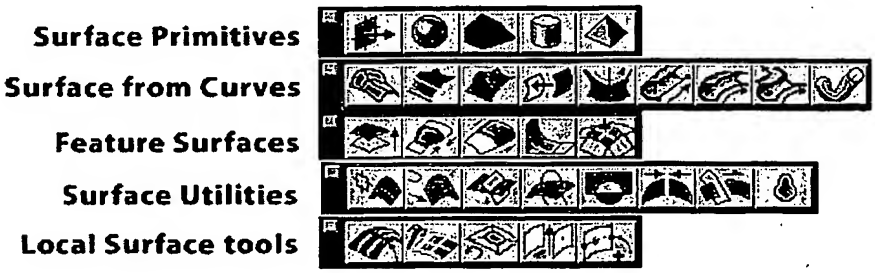
The palette will appear under the main tool palette on the left side of your screen.

You can save the open status of the palette and its location, simply by quitting your Designer Elements program. The next time you launch the program, the palette will be open and placed in the last location.

Each icon in the palette is the first in a subpalette of tools grouped together with a similar purpose. From top to bottom they include:

- **Surface Primitive** tools
- **Surface From Curves** tools

- **Feature Surface** tools
- **Surface Utility** tools
- **Local Surface** tools



Surface Primitive Tools

The Surface Primitives tool palette provides tools for quickly creating surfaces from simple shapes and parameters. This tool palette provides a means to create **Infinite Planes**, **Speheres**, **Blocks**, **Cylinders** and **Pyramids**. (For more information on these tools see chapter 16.)

Surface From Curves Tools

Use the Surfaces From Curves tools to construct surfaces built upon curves. This tool palette is home to the **Net**, **Ruled**, **Skin**, **Cover**, **Revolution** (lathe), **Extruded**, **Swept**, **Swept 2 rail** and **Pipe** surface tools.(For more information on these tools see chapter 16.)

Feature Surface Tools

The Feature Surface tool palette constructs surfaces built upon other surfaces. The tools in this palette include **Offset**, **Loft**, **Draft**, **Fillet** and **Tangent Cover Surface**. (For more information on these tools see chapter 16.)

Surface Utilites Tools

After you've constructed or imported surfaces into your model, you may need to refine or combine the surfaes with other entities. Your Designer Elements program provides several tools which make these tasks easy to complete. The Surface Utilities palette is home to tools such as **Entity Intersections**, **Projections**, **Joining**, **Trimming** and **Silhouette Curves** commands. (For more information on these tools see chapter 21.)

Local Surface Tools

This tool palette provides commands to modify the internal shape or definition of a surface. Example tools located in this palette include, **Surface Slope Matching**, **Rebuilding**, **Untrimming**, **Degree Elevation** and **Knot Insertion**. (For more information on these tools see chapter 21.)

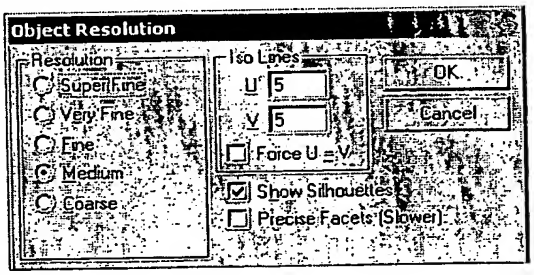
Surfaces, Selection and Display

You can select a surface by clicking anywhere within its boundary. However, your Designer Elements program will find the surface fastest if you select the surface's edge. A surface is displayed by its boundary edges and any existing interior holes. You can increase a surface's edge resolution with the *Change Resolution* command in the Edit menu.

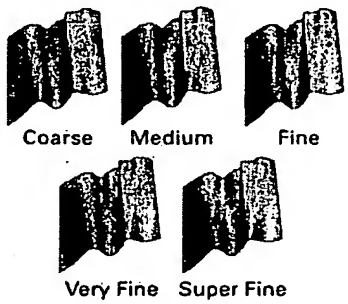
Changing an Object's Resolution

1. Select the object.
2. Choose ***Edit>Change Resolution.***

   The following Object Resolution dialog box appears.



You can choose one of five resolutions, Super Fine, Very Fine, Fine, Medium and Coarse. Select the desired resolution and click OK to close the dialog box. The object changes accordingly. The graphic illustrates each option.



3. Enter the desired number of Iso lines. Check the *Show Silhouettes* box if desired.

   The *# Iso Lines* allow you to control the *isopram* lines drawn for a surface. These Iso (isopram) lines are constant parameter curves that lie on a surface, typically defined in parameter space. The parameter space coordinate system uses *U* and *V* coordinates. A 0 (zero) in both fields turns off Iso lines. The appropriate U/V values may enhance the visual appearance of the surface at the expense of drawing speed. The letters, U and V are space coordinate identifiers (U = horizontal, V = vertical) that are the industry standard. The Force U=V check box automatically sets equal number of U and V lines. Entering a new value in one automatically changes the other isoline value.

   The *Silhouette* check box controls the silhouette edges of objects. Silhouette edges are view dependent and can cause a significant reduction in drawing speed. A check in the box turns the silhouette on.

4. Click OK to close the dialog box and save the new settings.

Surface Associativity

Surfaces are associative to the original curves used to create them. Changing the curve automatically changes the dependent surface. For example, if you create a skin surface from three arcs and change the radius of one of the arcs, the skin surface automatically updates to reflect the change. Use the ***Edit>Remove Links*** command to remove all associativity with the surface if you do not want curve changes to affect the surface. Another important issue to remember with the curve/surface associativity is that if the curve is deleted, the surface deletes as well. Since it is defined by the curve, it is no longer valid.

Cobalt and Xenon Only

Cutting and Pasting Surfaces

When you cut a surface into the paste buffer, two sets of data get stored. The first set of data is a Designer Elements program object that will precisely paste into the

current file or into another Designer Elements program drawing. The second set of data is a collection of display vectors in the bitmap format. You can paste this data into other Windows or Macintosh programs that support the clipboard.

Exporting Surfaces

Your Designer Elements program provides several methods for exporting surfaces into other applications. Our primary goal is for you to preserve as much data a possible during the export. Use the following export option order when exporting surfaces:

- ASIS (.SAT)
- Parasolids (.x_t) (Windows only)
- ProE (.g) (Windows only)
- STEP
- DXF with imbedded .sat
- IGES
- StereoLithography (.stl) (Mesh surfaces only)

Objects Types and Edit Objects

Every surface or mesh object you create with a Designer Elements program tool is defined by its own characteristics, and include geometric characteristics and attributes. These are displayed in the Edit Objects dialog box. While the characteristics listed in the Attributes tab are identical, the characteristics listed in the Geometry tab vary with each object.

Chapters 15 and 16 introduce you to the surface tools of your Designer Elements program. Included with every tool description is a list of the characteristics displayed for the object when Edit Objects is chosen. For information on using the *Edit Objects* command and the dialog box, see Chapter 24.

Mesh Creation Tools -

A mesh is a collection of planar elements typically with three or four sides. It is equivalent to the 3D Mesh element found in DXF files.

You can use meshes to model bodies that span areas and for calculating wetted areas and volume attributes. Although less accurate than surface modeling, mesh modeling uses less memory, and still provides considerable flexibility for those who prefer to design only with cross sections.

Unlike a surface, a mesh object is not defined by a mathematical formula but by nodes and 3D vertices. The resolution or precision of a mesh is determined by the number of vertices.

A mesh object can be used to create output to rapid prototyping machines and for compatibility with AutoCAD R12 files. A mesh object is helpful to people who want to control the exact number of facets created for each model.

Of the mesh tools available in your Designer Elements program, the **Mesh Curve** tool is the most powerful. With this tool, you can create sections and "skin" them together to obtain areas. Skins can be combined into bodies to calculate body area curves. Calculations of the area underneath the curve also yield the body's volume.

For each tool you can enter values in the Status Line to define a mesh, either before or after you create the mesh. If you enter the values after you select the tool but before you create the mesh, your first click in the drawing area automatically registers all Status Line values. If you enter values in the selected Status Line data field

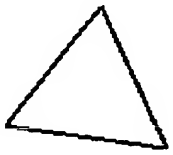
after creating the mesh and while the mesh is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the mesh to reflect the new values.

Your Designer Elements program supports four tools for creating mesh objects, **Mesh 3 Pts**, **Mesh 4 Pts**, **Mesh Curve** and **Mesh Extrude**. The topics explained in this chapter include:

- **Mesh 3 Pts** tool
- **Mesh 4 Pts** tool
- **Mesh Curve** tool
- **Mesh Extrude** tool
- Mesh and Rendering
- Mesh Notations

To access the **Mesh** tools go to ***Window>Mesh***.

Mesh 3 Pts Tool



The **Mesh 3 Pts** tool creates a flat plate triangular object from three points that you select.

Using the Mesh 3 Pts Tool

Tech Note:

To select a mesh object immediately after you create it, choose the **Selection** tool. Otherwise, you will continue to create new mesh objects.

1. Select the tool. The Message Line reads: *Mesh 3 Pts: Enter 3 locations for mesh.*

   If you would like to create the mesh in a plane different than the one currently set, change the plane before moving on to the next step.

2. Click three points to define the mesh boundary.

   The mesh is created.

There are no entries in the Status Line.

Geometric Characteristics

A 3 points mesh is created by picking the three desired points. A mesh is made up of the following characteristic according to the Edit Objects dialog box: the Defining Point for the X, Y and Z location.

Defining Point refers to the X, Y and Z location of the chosen point. The section below Defining Points lists the points with the X, Y and Z location. The selected point is displayed in the Defining Point fields. Each point can be edited individually using these fields.

This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the mesh and choose ***Window>Edit Objects*** or double-click on the mesh.

Mesh 4 Pts Tool

The **Mesh 4 Pts** tool creates a flat plate rectangular object from four points that you select.

Using the Mesh 4 Pts Tool

1. Select the tool. The Message Line reads: *Mesh 4 Pts: Enter 4 locations for mesh.*

   If you would like to create the mesh in a plane different than the one currently set, change the plane before moving on to the next step.

2. Click four points to define the mesh boundary.

   The mesh is created.

There are no entries in the Status Line.

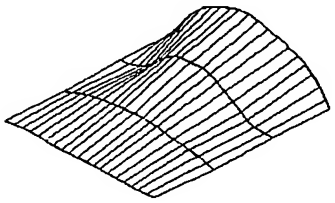
Geometric Characteristics

A 4 points mesh is created by picking the four desired points. A mesh is made up of the following characteristics according to the Edit Objects dialog box: the Defining Point for the X, Y and Z location.

Defining Point refers to the X, Y and Z location of the chosen point. The section below Defining Points lists the points with the X, Y and Z location. The selected point is displayed in the Defining Point fields. Each point can be edited individually using these fields.

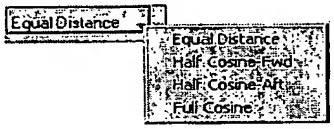
This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the mesh and choose ***Window>Edit Objects*** or double-click on the mesh.

Mesh Curve Tool



The **Mesh Curve** tool creates an *M* by *N* mesh from a collection of curves. M is the number of curves you select to define the mesh boundary. The mesh is created from the order in which you select them. N is number of segment lines you want displayed on the mesh. The graphic above shows a mesh curve with M = 4 (curves) and N = 20 (grid lines).

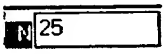
The **Mesh Curve** tool provides several options that control mesh spacing. You can pick them from the drop down list in the Message Line. The options include:



Equal Distance	Mesh points are spaced equally.
Half Cosine (Fwd)	Mesh points are spaced more closely at the forward end of the mesh, according to a half cosine.
Half Cosine (Aft)	Mesh points are spaced more closely at the back end of the mesh, according to a half cosine.
Full Cosine	Mesh points are spaced more closely at both ends of the mesh, according to a full cosine.

Using the Mesh Curve Tool

1. Select the tool. The Message Line reads: *Mesh Curve: Select curves in order of meshing [Shift=Extend].*
2. Select a mesh spacing option from the pull-down menu.
3. The Status Line displays the N data field which represents the number of mesh segments lines that will be distributed across the mesh.



If you want a different number of segments, enter the new value in the data field. The default value is 25.

4. Click two or more curves to define the mesh boundary. These curves should not be connected. If you need to select more than two curves, hold down the SHIFT key before you select the first curve.

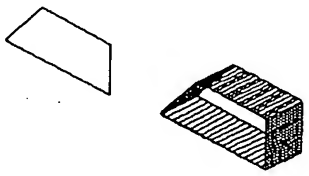
   Select the curves in the order that you want the mesh created.

Geometric Characteristics

A mesh curve is created by selecting the desired curves. A mesh curve is made up of the following characteristics according to the Edit Objects dialog box: the Distribution setting (Equal, Half Cosine (Forward), Half Cosine (Aft) or Full Cosine) and Segments (N) which make up the mesh.

This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the mesh and choose ***Window>Edit Objects*** or double-click on the mesh.

Mesh Extrude Tool



The **Mesh Extrude** tool sweeps a collection of curves along a vector, creating *N* mesh segments.

Using the Mesh Extrude Tool

1. Select the tool. The Message Line reads: *Mesh Extrude: Select objects to extrude [Shift=Extend].*
2. The Status Line displays data fields allowing you to enter the delta values for the extrusion and the number of N segments for the mesh.

3. Click one or more curves to sweep. If you need to select more than one curve, hold down the SHIFT key before you select the first curve.
4. If you want a different value for the N segments than currently listed, select the field and type the new value. After you selected the curves, The Message Line reads: *Mesh Extrude: Enter two points to extrude.*
5. Click two points to set the direction and the distance of the extrusion. The points do not have to be on the selected curves.

   The mesh is created.

If you want to modify the mesh while it's still selected, type the value in the appropriate field tabbing between fields as necessary. Press ENTER (Windows) or RETURN (Macintosh) and the mesh curve will be modified.

Geometric Characteristics

An extruded mesh is created by selecting the desired curves and the extrusion values that define the distance and direction. An extruded mesh is made up of the following characteristics according to the Edit Objects dialog box: Vertex Count, Facet Count and Segments. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the mesh and choose ***Window>Edit Objects*** or double-click on the mesh.

Mesh and Rendering

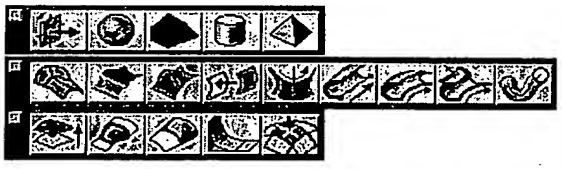
Mesh is displayed according to the Static Render setting in the Rendering Option dialog box. You can display the mesh as Wireframe, Flat, Gouraud, Gouraud w/ Edges, Phong, Phong w/Edges, Hidden and Hidden with Dimmed. See Chapter 33, "Rendering," for an explanation of these options.

Mesh Notations

The following notations will be helpful as you begin creating mesh objects.

- Mesh objects are not surfaces. They are defined by nodes or 3D vertices.
- Since mesh objects are not surfaces, you cannot use the **Surface Editing** tools contained in the third set of tools within the **Surface** tool palette.
- You can convert surfaces and solids into meshes and convert a closed mesh back into a solid.

Surface Creation Tools



Your Designer Elements program has 11 tools for creating surfaces. Every tool but the **Infinite Plane** tool can display a surface with *Iso Lines*. Iso (or isopram) lines are constant parameter curves that lie on a surface (defined mathematically in parameter space). U/V Iso lines refer to parameter space coordinates where U and V are letter identifiers. U Iso lines run perpendicularly to V Iso lines on a surface.

For each tool you can enter values in the Status Line to define a surface, either before or after you create the surface. If you enter the values after you select the tool but before you create the surface, your first click in the drawing area automatically registers all Status Line values. If you enter values in the selected Status Line data field after creating the surface and while the surface is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the surface to reflect the new values.

Many of the tools use a two step process. The first step after choosing the tool may be to select one or more objects. The second step is to perform the editing operation. Once you the complete the first step (which may involve holding down the SHIFT key to select multiple objects), you cannot select more objects to include in the same operation. To select additional objects, reset the tool and start again.

The tools are explained in the order of the palettes and include:

- **Infinite Plane**
- **Sphere Primitive**
- **Block Primitive**
- **Cylinder Primitive**
- **Pyramid Primitive**
- **Net Surface**
- **Ruled Surface**
- **Skin Surface**
- **Cover Surface**
- **Revolved Surface**
- **Sweep 2pts Surface**
- **Sweep 1 Rail Surface**
- **Sweep 2 Rail Surface**
- **Tube Surface**
- **Offset Surface**
- **Lofted Surface**
- **Draft Surface**
- **Fillet Surface**
- **Tangent Surface**

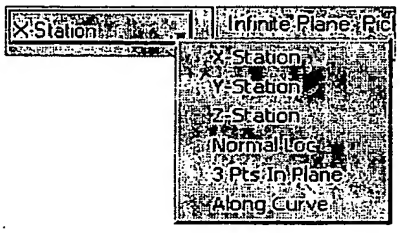
Infinite Plane Tool

The *Infinite Plane* is the simplest surface supported by your Designer Elements program. An infinite plane is defined by a location and a normal. The plane surface is useful for generating cross-sections through meshes, surfaces, or solids.

The infinite plane has six creation methods listed in the pull-down menu of the Message Line.

X-Station	Creates a plane with normal X=1, Y=0, Z=0.
Y-Station	Creates a plane with normal X=0, Y=1, Z=0.
Z-Station	Creates a plane with normal X=0, Y=0, Z=1.
Normal Location	Creates an arbitrary plane by specifying the location and normal.
3 Pts in Plane	Specifies a plane by three points. The normal is calculated from the three points.
Along Curve	Lays an evenly spaced series of planes along a selected curve. The plane's normals lie parallel to the curve.

The choices above are accessed from the Message Line, in the pull down pictured below.



After creating an infinite plane, it is represented on the screen by the following icon:

Using the Infinite Plane Tool for X-,Y- and Z-Station Planes

1. Select the **Infinite Plane** tool.
2. Pick *X-*, *Y-* or *Z-Station* from the drop down list on the Message Line. The Message Line reads: *Infinite Plane: Pick location for plane.*

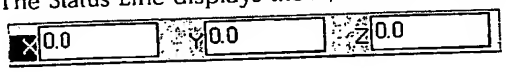
   If you would like to create the surface in a plane different from the one currently set, change the plane before moving on to the next step.

Tip:

You can determine the distance between two infinite planes by selecting the planes and choosing **Verify>Minimum Distance.**

3. Click the point on the screen where you want the plane. An icon for the plane appears at that location.

The Status Line displays the X, Y and Z location for the plane.



You can modify the location by typing new values in the appropriate fields. Tab to move from field to field. Pressing ENTER (Windows) or RETURN (Macintosh) and the plane location changes to reflect those values. All future changes must be made in the Edit Objects dialog box.

Using the Infinite Plane Tool to create a Normal and Location Plane

1. Select the **Infinite Plane** tool.
2. Pick *Normal Loc* from the drop down list on the message line. The Message Line reads: *Infinite Plane: Enter 3 points (1-Normal End, 2-Normal Start, 3-Loc).*

   If you would like to create the surface in a plane different from the one currently set, change the plane before moving on to the next step.
3. Click the point where you want the plane's normal to end, the point where you want the plane's normal to start, and the point where you want the plane to be placed.

There are no entries in the Status Line.

Using the Infinite Plane Tool to create a Three Point Plane

1. Select the **Infinite Plane** tool.
2. Pick *3 Pts In Plane* option from the drop down list on the Message Line. The Message Line reads: *Infinite Plane: Enter 3 points in the plane.*

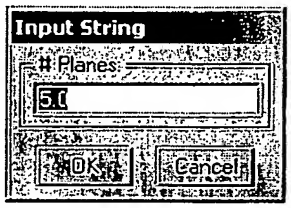
   If you would like to create the surface in a plane different from the one currently set, change the plane before moving on to the next step.
3. Click three points. The plane icon at the specified location.

There are no entries in the Status Line.

Using the Infinite Plane Tool to create a Plane along a Curve

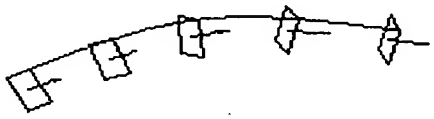
1. Select the **Infinite Plane** tool.

2. Pick *Along Curve* from the drop down list on the Message Line. The Message Line reads: *Infinite Plane: Pick curve to distribute planes along [Shift=Extend].*
3. Click a curve on the screen. The Input String dialog box appears.

   Enter the number of planes to place along the curve. Click OK to accept the change and close the dialog box. Click Cancel to close the dialog box without accepting the changes.

   The planes are created along a curve. The example here shows 5 planes along a curve.



If you select the curve, the Input String dialog box reappears allowing you to change the number of planes placed along the curve.

There are no entries in the Status Line.

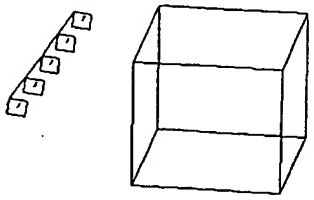
Infinite Plane Tool - Along the Curve Option and the Plane/Surface Intersection Tool

Referral:

See Chapter 22, "Surface Editing Tools," for information on using the **Plane/Surface Intersection** tool.

You can use the *Along Curve* option of the **Infinite Plane** tool with the **Plane/Surface Intersection** tool to quickly create cross sections through solid models.

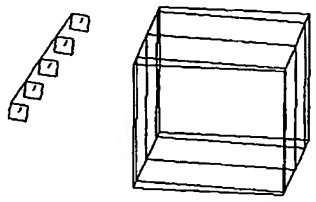
1. Select the **Infinite Plane** tool and select the *Along Curve* option.
2. Create the planes along a curve adjacent to the solid model.
3. Choose the **Plane/Surface Intersection** tool and pick the planes on the curve with which you want to cut the solid. The graphic here shows one example.



4. Select the solid. The **Plane/Surface Intersection** tool projects the planes through the solid. Your Designer Elements program puts each projected plane on a separate layer if you click the *AutoLayers* box on the Message Line before you create them.

5. The intersections displays.

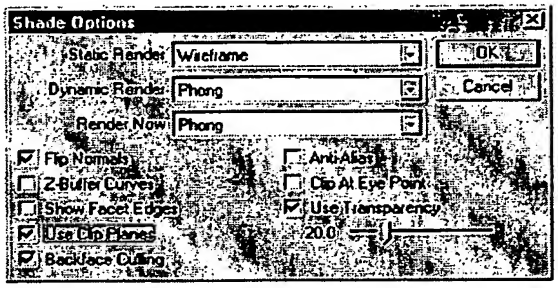
There are no entries in the Status Line.



Clipping Planes

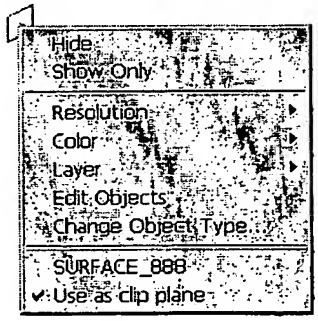
Dynamic display clipping planes provide a means to slice through the model and hide the display to either side of the plane.

In order to use an **Infinite Plane** as a **Clipping Plane** you must bring up the **Shade Options** dialog box located in the **View Menu** and select the "Use Clip Planes" option.



Using an Infinite Plane as a Clipping Plane

1. Select the Infinite Plane tool. The Message Line reads: *Infinite Plane: Pick location for plane.*
2. Place the Infinite Plane in your drawing.
3. Right-Click on the infinite plane and select the "Use a Clip Plane" menu option.



4. Use the **Selection** tool to dynamically drag the clip plane through the model.
5. To change the direction that the plane clips choose **Edit>Change Direction.**

This is also referred to as changing the surface normal or flipping normals.
To turn off the clipping plane option go to the **View>Shade Options** menu and deselect the Use Clip Plane option. By default this option is off.

Geometric Characteristics

An infinite plane is created by choosing points or curves as directed by the Message Line. An infinite plane is a surface made up of the following characteristics according to the Edit Objects dialog box: the Location for the X, Y and Z location and the Normal for the DX, DY and DZ location.

This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the plane and choose ***Window>Edit Objects*** or double-click on the plane.

Sphere Primitive

This tool creates a sphere surface using a center and radius point.

Using the Sphere Primitive Surface tool

1. Select the Sphere Primitive tool. The Message Line reads: *Sphere Primitive: Enter center and radius point for sphere.*
2. In the drawing area click the center point and radius point.

The Status Line contains X,Y and Z data fields for the sphere center point and dX, dY and dZ fields to reference distance and direction of the radius.

Block Primitive

The Block Surface Primitive creates a cubic or rectangular shaped object.

Using the Surface Block Primitive

1. Select the Block Primitive tool. The Message Line reads: *Block Primitive: Enter center and length points.*

2. In the drawing area click the center and length points for the block.

The first point you click is the center point. This is the bottom center of the resulting block. The second point you click controls not only length, but orientation of the block.

The Status Line contains X, Y and Z data fields for the block base center point plus dX, dY and dZ for the height. In addition, you can specify L and W for the length and width of the block.

Cylinder Primitive

This tool creats a cylindrical surface primitive.

Using the Surface Cylinder Primitive

1. Select the Cylinder Primitive tool. The Message Line reads: *Cylinder Primitive: Enter start and end points for cylinder.*
2. In the drawing area click the start and end points for the cylinder.

The first point you click is the cylinder base center point. The second point you click controls not only length, but orientation of the cylinder.

The Status Line contains X, Y and Z data fields for the cylinder base center point plus dX, dY and dZ for the height. In addition, you can specify D for the diameter of the cylinder.

Pyramid Primitive

This tool creates a pyramid surface.

Using the Surface Pyramid Primitive

1. Select the Surface Pyramid Primitive tool. The Message Line reads: *Pyramid Primitive: Enter start and end points for pyramid.*
2. In the drawing area click the start and end points for the pyramid.

The first point you click is the pyramid base center point. The second point you click controls the length and orientation of the pyramid.

The Status Line contains X, Y and Z data fields for the pyramid base center point plus dX, dY and dZ for the height. In addition, you can specify D for the pyramid base diameter and the #Sides for the number of sides

Surface From Curves Tools.

Net Surface Tool

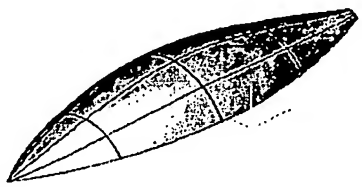
The **Net Surface** tool creates a surface defined by a grid of *M* rows and *N* columns, where M and N are letter identifiers referring to the number of rows or columns. Curves suitable as boundaries for net surfaces include arcs, circles, lines, splines, ellipses and conics. These curves cannot be grouped.

Tech Note:

A net surface is not the same as a mesh. Net surfaces are nurb surfaces. A mesh is defined by nodes or 3D vertices.

The curves may be in different planes. If the curves lie in the same plane, our Designer Elements program creates a simple planar surface within the curve boundary. If the curves lie out of plane, our Designer Elements program fits a curved net surface similar to a *Coons patch* mesh to the boundary curves. A Coons patch is a three or four sided nurb surface. The left graphic below shows four curves. The right graphic shows the resulting net surface with the referenced curves in gray.

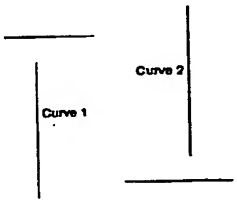
You can also specify a point entity for the start or end of the net surface, creating a degenerative surface.



Using the Net Surface Tool

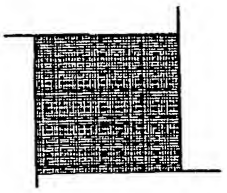
1. Select the **Net Surface** tool. The Message Line reads: *Net Surface: Pick (M) curves defining surface [Shift=Extend].*

2. Hold down the SHIFT key and pick two or more curves along one direction. The objects appear selected as in the graphic here (Curves 1 and 2).

   When you release the SHIFT key, the objects are deselected. The Message Line now reads, *Net Surface: Pick (N) curves defining surface [Shift=Extend].*

3. Hold down the SHIFT key once again, and pick two or more curves to complete the boundary for the surface.

   When you release the mouse button the net surface is created. The graphic here shows the net surface from the example above.

As the example shows here, the curves defining the surface do not have to be connected. Our Designer Elements program calculates the intersecting area of the selected curves and creates the net surface from that area.

There are no entries in the Status Line.

Geometric Characteristics

A net surface is created by choosing curves as directed by the Message Line. A net surface has no specific geometry characteristics according to the Edit Objects dialog box. There are tabs for both Display and Attributes.

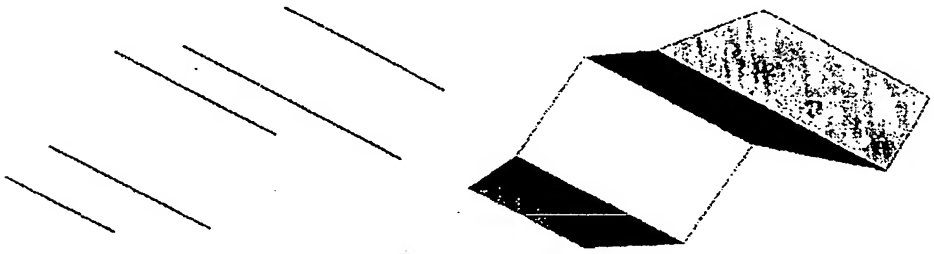
Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

Ruled Surface Tool

The ruled surface tool is used to create surfaces that are linear between the selected spans. Ruled surfaces are developable surfaces. Developable surfaces can be unrolled into flat patterns without stretching or shrinking.

Using the Ruled Surface Tool

1. Select the **Ruled Surface** tool. The Message Line reads: *Ruled Surface: Pick curves for defining surface. [Shift=Extend]*
2. If there are more than two curves being used for the ruled surface hold down the SHIFT key and select all of your curves. When you release the SHIFT key the surface is created



As the example shows, the curves defining the surface do not have to be connected. Our Designer Elements program calculates the intersecting area of the selected curves and creates the ruled surface

Geometric Characteristics

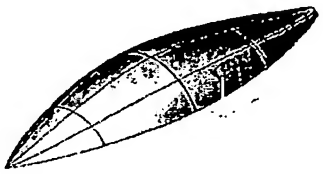
A ruled surface is created by choosing curves as directed by the Message Line. A ruled surface has no specific geometry characteristics according to the Edit Objects dialog box. There are tabs for both Display and Attributes.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

Tech Note:

Do not confuse the Skin Surface tool with the Cover Surface tool. The curves you select for this tool are used as cross-sections. They cannot be connected or intersecting. A warning message will appear if you choose invalid curves for the operation. To surface curves that are connected or closed use the **Cover Surface** tool.

Skin Surface Tools

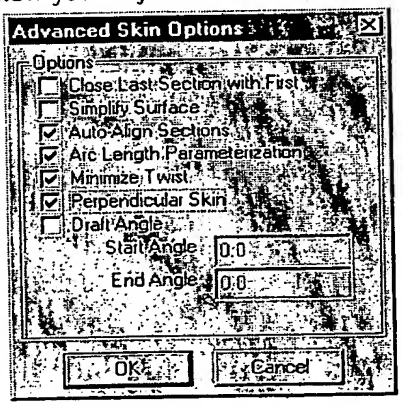


A skin surface fits a network of NURB patches to a collection of curves that act as *cross-sections* for the final surface. The curves that are skinned can be open, closed or grouped. Curves suitable for skinning include lines, arcs, circles, ellipses, conics and splines. You can create a standard skin surface or closed (tangent) skin surface. You can also specify a point entity for the start or end of the surface, creating a degenerative surface.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when creating a skin surface.

There are two tools in the Message Line, **Skin Curve** and **Skin with Guide Curves**.

There is also an Advanced Skin Options dialog box that gives you more control of how your object looks when skinned.



It contains the following options:

Close Last Section with First The close option may be used when the user needs to construct a body closed in one direction (v direction). The default is an open (not closed) body. The surface is continuous at each profile. If the user provides a set of closed profiles, the face normals of the skin or loft body point outside, away from the body material. When the user provides a set of open profiles, the surface normals of the skin surface is oriented along the surface normals and no attempt is made to change the surface normal orientation. When using this option you do not provide profiles in the same position.

Simplify Surface The simplify option simplifies the created surface to a conical surface, if applicable. If all of the cross sections lie on a conical surface (plane, cylinder, cone, sphere or torus), the conical surface is created instead. The value of 0.000001 is used to determine whether or not the cross section lies on an analytical surface (planar, conical, spherical or toroidal). The default is not simplified.

Auto-Align Sections The align sections option may be used to allow the skinning algorithm to align the direction of the curves in the selection list. Closed loops of wires can also be aligned. The default is aligned.

Arc Length Parameterization The arc length option is used to choose arc length or isoparametric parameterization of the skinning surfaces. In isoparametric parameterization the surface parameter in the v direction follows the cross section curves. In arc length parameterization the surface parameter follows lines of constant length. The default is arc length parameterization.

Minimize Twist The twist option may be used to minimize the twist of the surface produced. Twist minimization aligns closed curves such that the start of the second curve is aligned to the start of the first curve. Even if a body's shape is unaffected by twisting, a surface with a twist could produce unexpected

results when faceting and rendering. By default, twist minimization is on. Twist minimization is also an involved calculation that some users may not want to have carried out.

Perpendicular Skin

The take-off vector is a tangent vector going out of the starting edge or surface and into the skinned surface. The perpendicular option is used to specify the direction of the take-off vector, perpendicular to the edge. The default is in the loft direction because a perpendicular take-off vector can cause self-intersections to the surface.

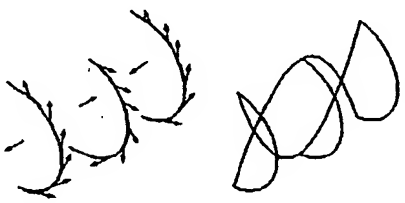
Draft Angle

Skinning with draft angles provides the ability to control the take-off vectors of the two outer skinning profiles. The "draft" angle is defined as an angle off the plane of the wire at every point along the skinning profile. In addition to the user supplying the angle itself, one may also supply a magnitude for the take-off vector. The draft angle and magnitude is constant for the entire profile. However, one may apply different draft angles to the two outer profiles. In addition skinning with draft angles supports open and closed profiles and skinning to a point. When skinning to a point the algorithm constructs its own normal vector. The outer profiles must be planar when not degenerate. Use the Edit Object dialog box to access the skin magnitude.

Standard Skin Surface

An open skin surface is a surface that has open ended in all directions. You select the curves in the order that you want them skinned. When you select the successive curves (or *n#* sections), you create a surface between the curves. You are not surfacing the individual curves. If you want to surface curves connected end to end use the **Cover Surface** tool. See a later section in this chapter.

The curves must have the same direction. If the curves have different directions the resulting surface will twist from section to section, as shown here.



To avoid creating twisted surfaces, check the curves' directions by choosing ***Verify>Direction***. If the curves do run in different directions you can adjust or redraw them before creating the surface. The graphic here shows curves with the same direction and the resulting surface.

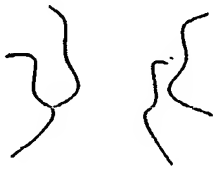
This tool also provides an option for creating closed tangents.

Tech Note:

The term, *N# sections*, refers to the curve sections you select to create a surface. The letter N is the generic identifier.

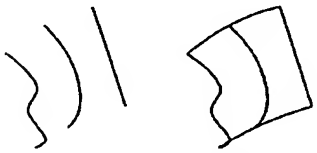
Closed Skin Surface

A closed or tangent skin surface is continuous end to end. When you use this option, your Designer Elements program automatically closes the skin such that the first section equals the last section making the slopes along the edge tangent continuous. The graphic here shows spline curves used to create a closed skin surface with the **Skin with Guide Curves** tool.



When these splines are skinned normally, the left model here results. When skinned using the closed skin option, the right model results.

Skin Curve Surface Tool



This tool creates a skin surface between two or more curves. The surface edges are defined by the limits of the selected curves. You can create a standard skin surface or a closed skin surface. The figure here shows a open skin surface created from three curves.

Creating a Standard Skin Surface

1. Check curve directions by choosing ***Verify>Direction***. If all of the curves do not run in the same direction, adjust them so that they do.
2. Select the **Skin Surface** tool. The Message Line reads: *Skin Surface: Pick curves for defining surface [Ctrl = Advanced* (Windows) or *Option* (Macintosh) *Shift=Extend].*

3. Select the **Skin Curve Surface** tool (the left tool) in the Message Line.
4. Select the curves in the order that the surface is to be skinned. To select more than two curves hold down the SHIFT key. You cannot use a selection fence to select the curves.

There are no entries in the Status Line.

Creating an Closed Skin Surface

1. Check curve directions by choosing ***Verify>Direction***. If all of the curves do not run in the same direction, adjust them so that they do.
2. Select the **Skin Surface** tool. The Message Line reads: *Skin Surface: Pick curves for defining surface [Ctrl = Closed* (Windows) or *Option* (Macintosh) *Shift=Extend].*
3. Select the **Skin Curve Surface** tool (the left tool) in the Message Line.
4. (Windows) Select the curves in the order that the surface is to be skinned. To select more than two curves hold down the SHIFT key. (You cannot use the selection fence to select the curves.) After you select the last curve and while still holding down the SHIFT key, hold down the CTRL key. Release the SHIFT key and then the CTRL key.

   (Macintosh) Hold down the OPTION key and select the curves in the order that

the surface is to be skinned. To select more than two curves hold down the SHIFT+OPTION keys. You cannot use the selection fence to select the curves.

The tangent surface is created.

There are no entries in the Status Line.

Skin with Curve Guides Surface Tool

This tool creates a skin surface between two or more curves using curves as guides to define the edges. Guide curves give you control over the skin surface. You can use one or more guide curves to influence the surface.

In the left graphic below, the profile curves are black and the guide curves are gray. The right graphic shows the skinned surface.

Guide Curve Rules

- Curves can go in any direction and need not be consistent with the others.
- Curves cannot loop and must be "well-behaved."
- The curves must connect with each profile that you want to use for surface creation.

Note: Guide curves only affect the geometry of the surface created between the profiles to which the guide curves are attached. Also, if the guide curve passes through vertices, the two adjoining surfaces follow the curve profile.

You can create a standard skin surface or a closed skin surface.

Creating a Standard Skin Surface

1. Check curve directions of the profile curves by choosing ***Verify>Direction***. If all of the curves do not run in the same direction, adjust them so that they do.
2. Create one or more guide curves.

3. Select the **Skin Surface** tool. The Message Line reads: *Skin Surface: Pick curves for defining surface [Ctrl = Advanced* (Windows) or *Option* (Macintosh) *Shift=Extend].*

4. Select the **Skin with Guide Curves Surface** tool (the right tool) in the Message Line.
5. Select the curves in the order that the surface is to be skinned. To select more than two curves, hold down the SHIFT key. You cannot use a selection fence to select the curves.

   The Message Line now reads, *Skin Surface: Pick guide curves [Shift = Extend].*
6. Select the guide curves.

   The skin surface is created.

There are no entries in the Status Line.

Creating a Closed Skin Surface

1. Check curve directions by choosing ***Verify>Direction***. If all of the curves do not run in the same direction, adjust them so that they do.
2. Create one or more guide curves.
3. Select the **Skin Surface** tool. The Message Line reads: *Skin Surface: Pick curves for defining surface [Ctrl = Advanced* (Windows) or *Option* (Macintosh) *Shift=Extend].*
4. Select the **Skin with Guide Curves Surface** tool (the right tool) in the Message Line.
5. (Windows) Select the curves in the order that the surface is to be skinned. To select more than two curves hold down the SHIFT key. (You cannot use the selection fence to select the curves.) After you select the last curve and while still holding down the SHIFT key, hold down the CTRL key. Release the SHIFT key and then the CTRL key.

   (Macintosh) Hold down the OPTION key and select the curves in the order that the surface is to be skinned. To select more than two curves hold down the SHIFT+OPTION keys. You cannot use the selection fence to select the curves.

   The Message Line now reads, *Skin Surface: Pick guide curves [Shift = Extend].*
6. Select the guide curves.

The tangent surface is created.

There are no entries in the Status Line.

Geometric Characteristics

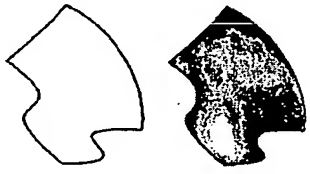
A skin surface is created by choosing curves as directed by the Message Line. A skin surface has no specific geometry characteristics according to the Edit Objects dialog box. There are tabs for both Display and Attributes.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting. Smart, Off or On.

Cover Surface Tool

The **Cover Surface** tool creates a surface from curves connected end to end. The tool tip calls these curves *n-sided*. Curves suitable as boundaries for covered surfaces include arcs, circles, lines, splines, ellipses, conics and grouped curves.

The curves may be in different planes. If the curves lie in the same plane our Designer Elements program creates a simple planar surface within the curve boundary. If the curves lie out of plane, our Designer Elements program fits a *Gregory surface* to the boundary curves. A Gregory surface is a nurb surface with five or more sides.

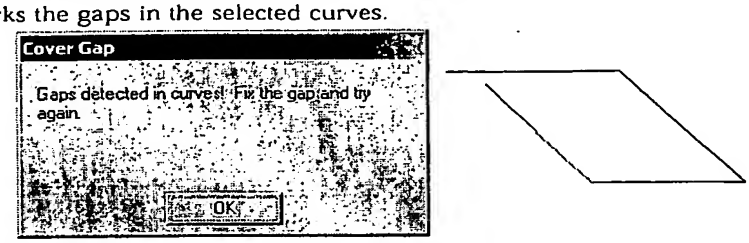


Using the Cover Surface Tool

1. Select the **Cover Surface** tool. The Message Line reads: *Cover Surface: Pick curves for cover surface [Shift=Extend].*
2. If you want to select more than one curve hold down the SHIFT key before selecting the first curve.
3. Select the curves. Do not worry about the order you select them.

   The cover surface is created.

If not all of the curves that you select connect end to end, as shown in the figure below, our Designer Elements program warns you with a Cover Gap dialog box and marks the gaps in the selected curves.



Either move the endpoints together or redraw the curves so that they connect.

There are no entries in the Status Line.

Geometric Characteristics

A cover surface is created by choosing curves as directed by the Message Line. A cover surface has no specific geometry characteristics according to the Edit Objects dialog box. There are tabs for both Display and Attributes.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On

Revolved Surface Tool

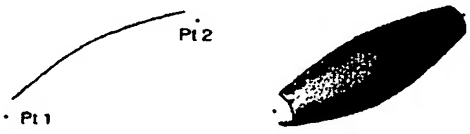
The **Revolve Surface** tool allows you can create a surface by revolving a set of curves (called the profile) around a central axis.

In the Message Line, you have a subpalette displaying two tools for revolving profiles: **Revolved Surface about Two Points** and **Revolved Surface about a Curve**.

If you revolve curves connected at their endpoints the resulting revolution creates one surface.

Revolved Surface about Two Points Tools

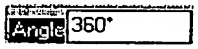
This tool allows you to revolve a curve about two points that you choose. The direction of the Angle of Revolution is determined by the order in which you pick the points that define the axis. Use the Right Hand Rule to determine the direction that the surface will revolve.



See Chapter 2 for more information on the Right Hand Rule.

Using the Surface of Revolution Tool - About Two Points

1. Select the **Revolved Surface** tool.
2. Select the **Resolve Surface About Two Points** tool in the Message Line. The Message Line reads: *Revolved Surface: Pick surface to revolve [Shift=Extend].*
3. Enter the angle through which to revolve the curve in the Angle data field of the Status Line.



4. Select the curve or curves to revolve.

   The Message Line directs you to enter two points for lathe (or revolution) axis.
5. Click two points to define the Axis of Revolution.

   If the Axis of Revolution crosses any of the profile curves this error message appears:

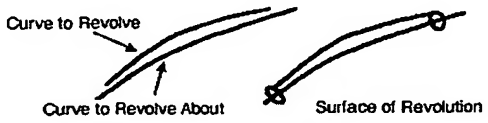
   Pick a new non-intersecting axis to complete the surface of revolution.

   The surface is created.

You can change the Angle while the surface is still selected by highlighting the Angle field, entering a new value and pressing ENTER (Windows) or RETURN (Macintosh). The surface adjusts accordingly. Any future modification to angle must be made through the Edit Objects dialog box.

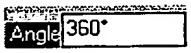
Revolved Surface about a Curve Tool

This tool allows you to revolve a curve about an existing curve that you choose. The direction of the angle of revolution is determined by the direction in which the axis of revolution curve was drawn. Use the Right Hand rule to determine the direction for the revolution. See Chapter 2 for more information about the Right Hand Rule.



Using the Revolved Surface Tool: About a Curve

1. Select the **Revolved Surface** tool.
2. Select the **Resolve Surface About Curve** tool in the Message Line. The Message Line reads: *Revolved Surface: Pick surface to revolve [Shift=Extend].*
3. Enter the angle through which to revolve the curve in the Angle data field on the Status Line.



4. Select the curve or curves to revolve.
5. Select the curve to use as the Axis of Revolution. If the Axis of Revolution crosses any of the profile curves and error message appears. Choose another curve or adjust the curve as required.

   The surface is created.

You can change the Angle while the surface is still selected by highlighting the Angle field, entering a new value and pressing ENTER (Windows) or RETURN (Macintosh). The surface will adjust accordingly. Any future modification to angle must be made through the Edit Objects dialog box.

Geometric Characteristics

A revolved surface is created by choosing a curve to revolve and the axis of revolution. A revolved surface has the following geometry characteristics according to the Edit Objects dialog box: Rot. Angle, End 1 (X, Y and Z values) and End 2 (X, Y and Z values).

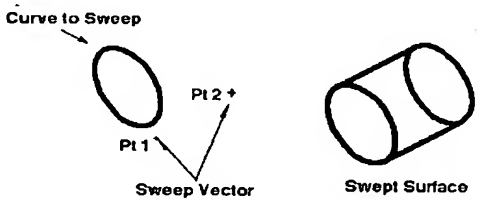
Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A

grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

Sweep Surface Tool - Along 2 Points

The **Sweep Surface** tool sweeps a curve along a path defined by the user. The path chosen depends on the **Sweep Surface** tool used. This Designer Elements program automatically aligns, orders and sets the direction of your curves for the sweep path.

This tool sweeps a curve along a two point vector. The sweep surface starts from the curve and moves in the direction and distance indicated by two points that you choose. The resulting surface is associative to the original sweep curve. Change the curve and the surface automatically updates.

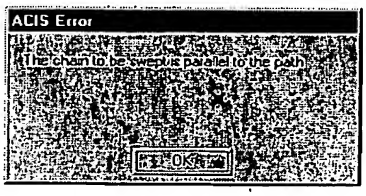


You can check information on the sweep using commands in the Verify menu. You can sweep grouped curves with this tool.

Using the Sweep Surface Tool: Along 2 Points

1. Select the **Sweep Surface Along 2 Points** tool. The Message Line reads: *Pick curve(s) to sweep [Shift=Extend].*
2. Select the curve that you want to sweep. The Message Line reads: *Sweep 2 Pts Surface: Enter two points for sweep direction.*
3. Click two points to define the vector. The order you click the points specifies the direction of the sweep.

   The vector cannot run in the same direction as the curve. If you define a vector in that direction this message appears:

   Select the curve again and click two other points to define the vector.

   The sweep surface is created.

Tip:

To sweep a curve at an angle not perpendicular to the curve, change the view to specify the sweep points. Example: If you draw a rectangle in the top plane, change the view to front and specify the points at the desired angle.

You can change the values in these data fields while the surface is still selected by highlighting the desired field and entering in a new value. Use the Tab key to move from field to field. Press ENTER (Windows) or RETURN (Macintosh) and the surface will adjust accordingly. Any changes you want to make after this must be made in the Edit Object dialog box.

The Status Line displays delta values for the X, Y and Z coordinates of the vector and the distance of the sweep.

Geometric Characteristics

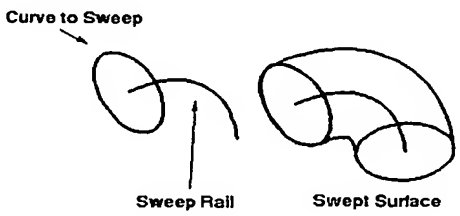
A swept surface is created by choosing a curve to sweep and the points for the two points defining the vector. A swept surface along two points has the following geometry characteristics according to the Edit Objects dialog box: Distance and Vector (dX, dY and dZ values).

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

Sweep Surface Tool - Along 1Rail

This tool creates a surface by sweeping an open, closed or grouped curve along another curve. The resulting surface is associative to both the path and the sweep curve.

Modifying either automatically updates the associative sweep surface. You can only sweep one curve at a time.



Using the Sweep Surface Tool:Along 1 Rail

1. Select the **Sweep Surface Along 1 Rail** tool. The Message Line reads: *Sweep 1 Rail Surface: Pick curve to sweep [Shift=Extend].*

2. Click the sweep curve. The Message Line reads: *Sweep 1 Rail Surface: Pick sweep axis [Shift=Extend].*
3. Click the rail curve. The surface is created.

There are no entries in the Status Line.

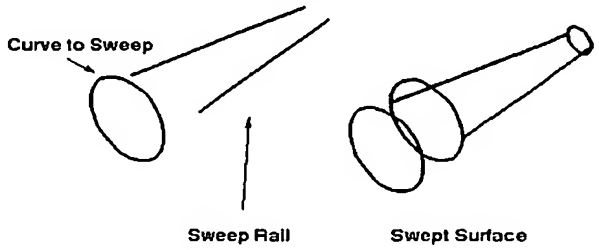
Geometric Characteristics

A swept surface is created by choosing a curve to sweep and a curve that acts as the rail for the sweep. A swept surface along one rail has no specific geometry characteristics according to the Edit Objects dialog box. There are tabs for both Display and Attributes.

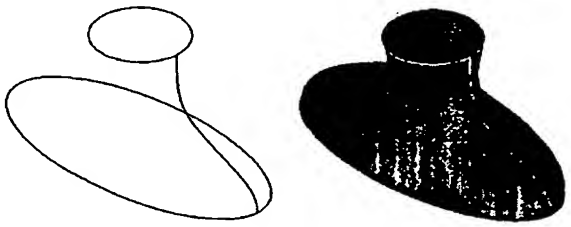
Display contains settings for Iso lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

Sweep Surface Tool - Along 2 Rails

The sweep along two rails method creates a swept surface by sweeping an open, closed or grouped profile between two rail curves. A rail can also be closed, back tracking over itself like a circle. This tool supports the maintaining of the profile height along the rail by holding down the CONTROL (Windows) or OPTION (Macintosh) key. This tool does not support multiple or grouped curves for the rails. A simple two rail sweep is shown in the graphic here.



As mentioned, you can also use closed curves or curves that backtrack as the rails. In the left graphic below, the top circle and bottom ellipse are the rails and the spline is the curve being swept. The right graphic shows the swept surface.



The rail curves define the orientation and scale of the swept surface as it moves between them. The resulting surface is associative to both rails and the sweep curve. Modifying any of the three will automatically update the associative sweep surface.

Using the Sweep Surface Tool: Along 2 Rails

1. Select the **Sweep Surface** tool.
2. Select the **Sweep Surface Along 2 Rails** tool in the Message Line. The Message Line reads: *Sweep 2 Rail Surface: Pick curves or group to sweep. [Ctrl or Option = Maintain Height][Shift = Extend]*
3. Click the profile you want to sweep and then choose the two rails for the sweep path.

   A swept surface is created.

   Hold down the Ctrl or Option key if you want the profile height to be maintained.

There are no entries in the Status Line.

Geometric Characteristics

A swept surface is created by choosing curves or grouped profiles that act as the rails for the sweep. A swept surface along two rails has no specific geometry characteristics according to the Edit Objects dialog box. There are tabs for both Display and Attributes.

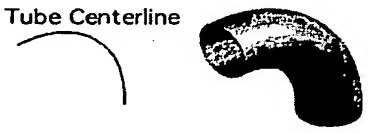
Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

Sweep Rules

- You must double-click the surface to display the Edit Objects dialog box. If you pick the surface and the original sweep curve with ***Edit>Select All***, or by dragging a selection fence, the Edit Objects dialog box will list only the attributes of the two entities and will omit the sweep surface's geometry information. Therefore, always select the surface by double-clicking it. With the second click, the Edit Objects dialog box appears automatically.
- If you want to perform a helical sweep, the helix must originate in this Designer Elements program. Imported helix curves do not follow the helix curve law and the sweep operation requires this for a proper sweep.
- If you sweep curves connected at their endpoints, the resulting sweep creates one surface.

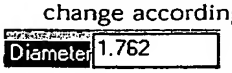
Tube Surface Tool

With the **Tube Surface** tool you can quickly and efficiently create a tube or pipe along a reference curve. This tool support grouped curves. You must specify a radius small enough to prevent a self-intersecting surface.



Using the Tube Surface Tool

1. Create a curve that would act as the center line for the tube.
2. Select the **Tube Surface** tool. The Message Line reads: *Tube Surface: Pick tube center line [Shift=Extend].*
3. Click the curve for the tube's center line.
4. Enter a new tube radius in the data field on the Status Line, if required. Press ENTER (Windows) or RETURN (Macintosh) and the radius of the tube will change accordingly.



If you want to specify a tube radius before the tube is created, type the value in the Radius data field and then select the curve and the tube surface will be created with the specified radius.

Geometric Characteristics

A tube surface is created by choosing a curve that will act as the center line for the tube. A tube surface has the following geometry characteristic according to the Edit Objects dialog box: Diameter of the tube.

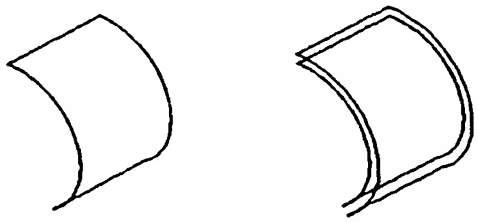
Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

Feature Surface Tools

Offset Surface Tool

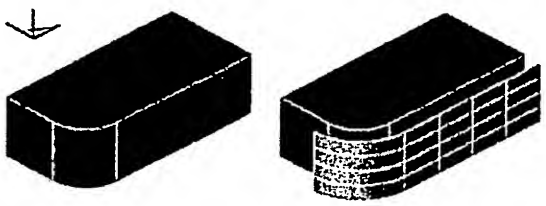
With the **Offset Surface** tool you can make a new surface based on an existing or parent surface, offset by an amount that you choose. You can also offset an infinite plane.

The offset surface is created by first selecting the parent surface and specifying the offset surface. Offset surfaces maintain a constant distance normal to the parent surface. The direction of the offset is the surface normal direction. You can specify negative offsets. Do not offset a surface a distance greater than the smallest radius of curvature or you can create a degenerate surface.



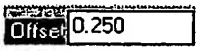
If you specify a zero offset, this Designer Elements program creates an associative copy of the selected surface at the same location.

You can also offset solids faces. You may single select a particular face or box select the entire solid to offset all of its faces. A positive offset for a solid, creates a surface outside the solid. A negative offset creates a surface towards the inside of the solid.



Using the Offset Surface Tool

1. Select the **Offset Surface** tool. The Message Line reads: *Offset Surface: Pick surfaces to offset [Shift=Extend].*
2. Enter an Offset distance in the data field of the Status Line.



3. Click the parent surface. A duplicate surface appears at the offset distance.

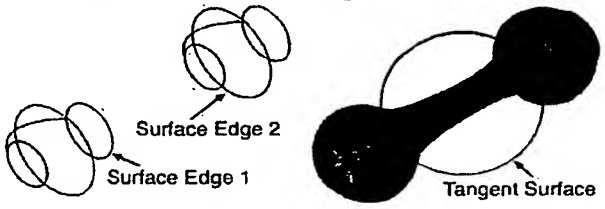
Geometric Characteristics

An offset surface is created by choosing the parent surface and specifying an offset distance. An offset surface is composed of the offset value according to the Edit Objects dialog box. There are tabs for both Display and Attributes.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

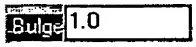
Lofted Surface Tool

The **Lofted Surface** tool creates a smooth, transitional surface between surfaces edges and any number of interior non-surface edges. To create a tangent surface select the edge of the start surface and the edge of the end surface.



Using the Lofted Surface Tool

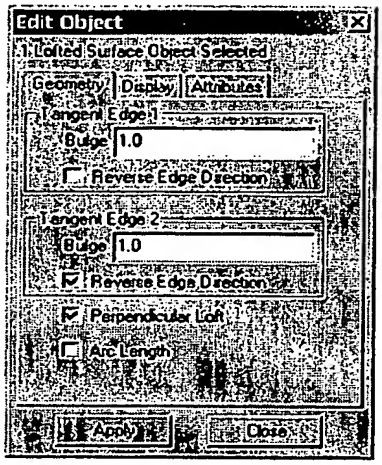
1. Select the **Lofted Surface** tool. The Message Line reads: *Lofted Surface: Pick edges for lofted surface.[Shift=Extend].*
2. Enter a new bulge factor in the Bulge data field in the Status Line, if desired. The bulge factor controls the magnitude of the slope of the surface edges.



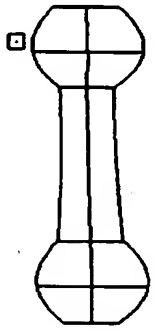
3. Click an edge on each surface to be joined by the tangent surface. When you create a tangent surface, the screen shows only one connecting line. Choose ***View>Shade Now*** to see the tangent surface. If *Shade Now* is set to anything other than wireframe in the Shade Options dialog box, your lofted surface appears in the chosen mode.

Once created, the lofted surface can be modified in the Edit Objects dialog box accessed through the **Design Explorer**. The start bulge factor controls the magnitude of the slope from the starting edge, whereas the end controls the slope at the

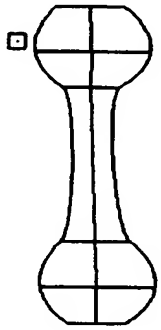
end curve. The **reverse Edge Direction** options are provided to fix twisting of the Tangent surface resulting from different edge directions.



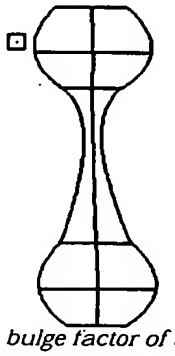
The figure shows two bulb surfaces joined with a tangent surface using different bulge factors. The left figure uses a 0.25 bulge factor; the center figure uses a 10.0 bulge factor; the right figure uses a 3.0 start bulge factor and a 0.5 end bulge factor.



A bulge factor of .25 is used at both ends.



A bulge factor of 10 is used at both ends.



A bulge factor of 3 is used at the top end and 0.50 factor at the bottom end.

Geometric Characteristics

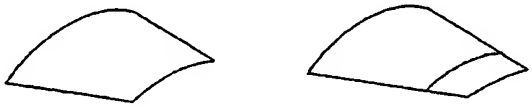
A lofted surface is created by choosing the two connecting surfaces. You may edit the bulge factor for the lofted surface edges in the Edit Objects dialog box. Tangent Edge 1 controls the first edge chosen and Tangent Edge 2 controls the last edge chosen.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

Draft Surface Tool

The **Draft Surface** tool extends a surface to create a new surface attached to the end of the original surface. This Designer Elements program provides data fields for angle and length that give you control over how the surface is generated. Entering 0° in the Angle data field extends the surface tangent to the original. Entering 90° extends the surface perpendicularly.

You have two options to set the new surface tangency; *Default* and *Align Curve*. The default option is automatic when using this tool. It creates a draft surface aligned with the original surface derivatives, based on the product of the surface tangent and normal. The graphic below shows the original geometry and the resulting draft surface.



By holding down the CTRL key (Windows) or OPTION key (Macintosh) you choose the *Align Curve* option listed in the Message Line. When you use this option a draft surface is created based on the product of the alignment curve tangent and the sur-

face normal. The graphic below shows the original geometry and the resulting draft surface.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when creating a draft surface.

Using the Draft Surface Tool

1. Select the **Draft Surface** tool. The Message Line reads: *Draft Surface: Select curve on surface [Ctrl=Align Curve].*
2. Enter a new angle and/or length in the data fields in the Status Line, if you so desire.

3. Click a curve (edge) on the surface to draft or extend the surface. When you select the edge, the whole surface is highlighted until this Designer Elements program creates the draft.

You can change both the angle and the length of the Draft by entering new values in the data field while the geometry is still selected. Pressing ENTER (Windows) or RETURN (Macintosh) will make the change. Any other changes must be made in the Edit Objects dialog box.

Geometric Characteristics

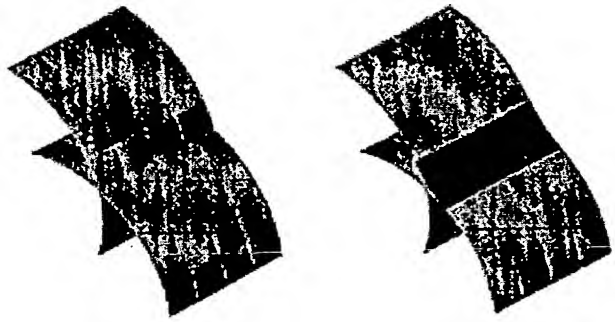
A draft surface is created by choosing a curve to draft and entering the desired angle and length of the draft. A draft surface has the following geometry characteristics according to the Edit Objects dialog box: Angle and Length.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

Fillet Surface Tool

The **Fillet Surface** tool blends two surfaces that share an edge or intersection. This tool does not extend surfaces.

When filleting two intersecting surfaces, there are four places where the fillet can be placed. The fillet is created where the surface directions are positive for both surfaces. The surface direction can be determined by selecting and choosing ***Verify>Direction***. You can change the direction by choosing ***Edit>Change Direction***. The left graphic here shows two intersecting surfaces. The right graphic shows the surfaces blended.



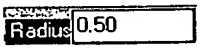
Using the Fillet Surface Tool

1. Select the **Fillet Surface** tool. The Message Line reads: *Fillet Surface: Pick two surfaces to blend [Shift=Extend].*
2. Select the two intersecting or connected surfaces that share an edge

   The surfaces are filleted.

If you want to change the fillet radius, type the value in the Radius data field in the Status Line and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the Radius for the blend.



Geometric Characteristics

A fillet surface has the following geometric characteristic according to the Edit Objects dialog box: Radius of the fillet.

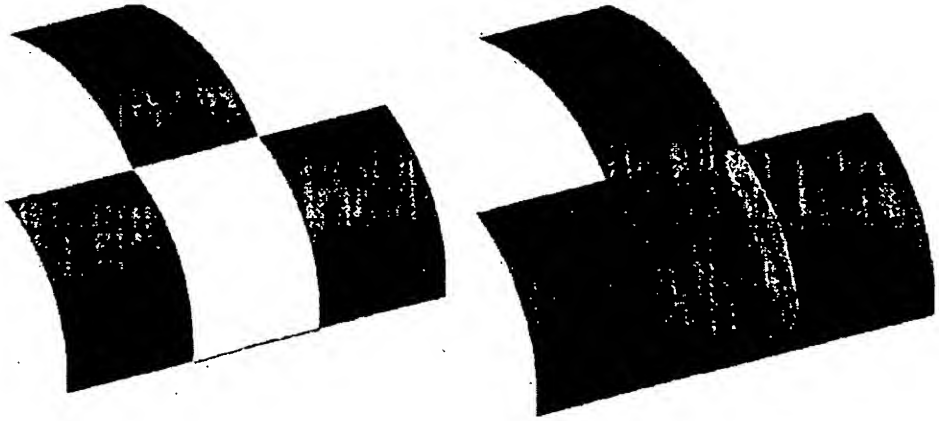
The Display page of Edit Objects contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

Tangent Cover Surface Tool

The **Tangent Cover** surfaced creates a surface that is tangent to four neighboring surfaces. The surfaces must form a closure. You may also select curves for some of the boundaries however if you have a choice, use surface edges for the boundary of the tangent cover surface.

Using the Tangent Cover Surface Tool

1. Select the Tangent Cover Surface tool. The Message Line reads: *Tangent Cover Surface: Pick four edges/curves for cover surface.*
2. Select the surface edges and or curves. Select them inhead to tail fashion. The tangent cover surface is created.



Note: The first surface selected is the driving surface. Changing the order of the surface selection can change the final results.

Introduction to Solid Modeling

A solid model is an unambiguous 3D representation of an object, composed of selectable faces. A solid model is unambiguous because their mathematical descriptions completely define the inside and outside of a 3D object.

Unlike wireframes and surfaces which define lengths and areas, solids accurately provide volume, mass, moments of inertia, centroids and interference information. Solids also generate stereolithography models useful for demonstrating prototype concepts or for creating forms for molds or castings.

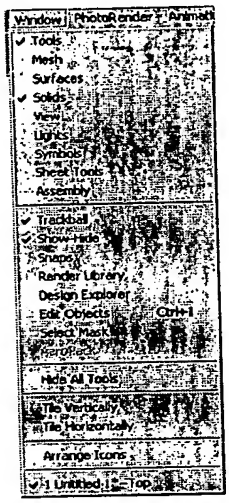
Our Designer Elements program is based on Spatial Technologies' ACIS Geometry Engine. ACIS provides a precise boundary representation of solids suitable for design projects sensitive to accuracy. In ACIS, linear and quadric geometry is exactly represented analytically while free-form geometry is represented as Non Uniform Rational B-Splines (NURBs). Unlike faceted solid modelers, ACIS maintains a high level of accuracy as you perform boolean operations or add features. Because the ACIS engine is so accurate, parts modeled with it are suitable for applications that demand precision, like numerical control. Our Designer Elements program provides several sets of tools with which to create or modify solids: Primitives, Profiles, Features and Editing.

The topics explained here include:

- Solid Modeling tools
- Drafting Assistant and Solids
- Solids, Selection and Display
- Solid Associativity

- Solids and Instances
- Cutting and Pasting Solids
- Exporting Solids
- Object Types and Edit Objects

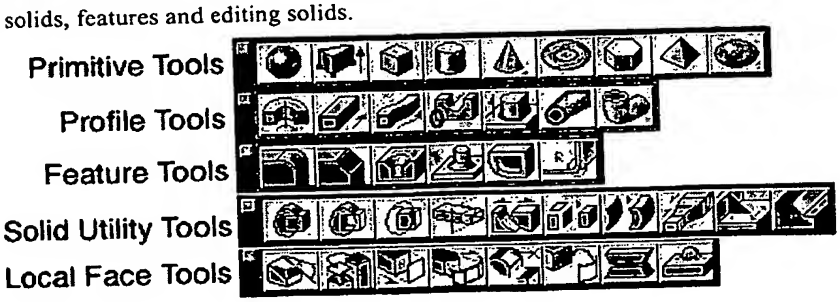
Solid Modeling Tools



The solid modeling tools are accessible from the Solids tool palette. The Solids tool palette displays automatically when you first launch your Designer Elements program. To display the palette, choose ***Window>Solids***. Selecting the Solids option again from the Window menu or the close box in the its palette's title bar, closes the palette.

The palette will appear under the main tool palette on the left side of your screen.

The five solid modeling subpalettes contain groups of tools for creating primitives, profiled solids, features and editing solids.



Primitives

A primitive is a simple shape such as a sphere, slab, block, cylinder, etc.

The Primitives subpalette contains the following tools: Sphere, Slab, Block, Cylinder, Cone, Torus, Prism, Pyramid and Ellipsoid. Chapter 18 explains these tools.

Profiles

A profile is a curve or collection of curves that enclose an area. You can create a solid from a profile by revolving, extruding, or sweeping a curve or pipe profile along a path.

The Profiles subpalette's contains the following tools: Lathed, Extrude, Sweep, Cutout, Protruded Feature, Skinned Solid and Pipe Solid. Chapter 19 explains these tools.

Features

A feature simplifies building or modifying a solid by constructing common design elements with a single command.

The Features subpalette's contains the following tools: Blends (constant, linear, variable), Chamfer (constant, angle, linear) Holes (simple hole, counter bore, counter sink), Boss, Shell and Bend. Chapter 22 explains these tools.

Solid Utility Tools

As you create your geometry you may want to perform various operations on them such as splitting a solid or placing a draft. The Solid Utility tools provide you with various ways of editing solid objects.

The Solid Editing tool subpalette contains the following tools: Boolean (Union, Subtract, Intersect), Trim, Split, Stitched, Thicken, Lofted, Rib Feature and Lip Feature (Remove and Add). Chapter 23 explains these tools.

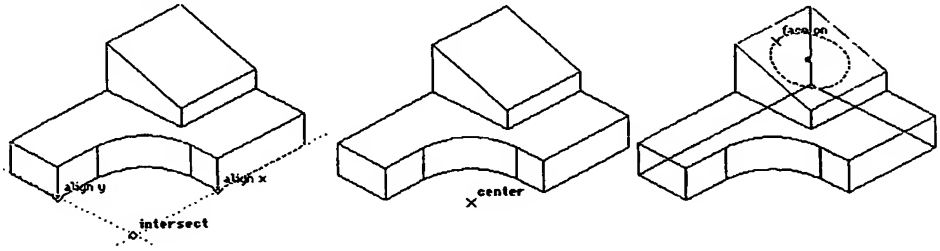
Local Solid Face Tools

As you create or import geometry you may want to perform various operations on it. The Local Solid Face Tools provide a means to modify individual faces of your solid geometry.

The Local Solid Face subpalette contains the following tools: Draft Solid, Match Face, Move Face, Offset Face, Remove Face, Replace Face, Parting Line and Deform Face (by disatance, to a point or to a curve). Chapter 23 explains these tools.

Drafting Assistant and Solids

The Drafting Assistant recognizes a variety of intelligent snap locations for solids including all hard vertices, user supplied points (grips), hole centers, cylinder centers, alignments, alignment intersections and fillet centers. The Drafting Assistant works the same for a solid as it does for a curve.



Solids, Selection and Display

You can select a solid by clicking on an edge or within a face. Selecting the solid on its edge is considerably faster than selecting within a face (our Designer Elements program has to cast a ray to see if it pierces the solid).

Displaying Solid Edges

Solids display three types of edges, Hard, Silhouette and Smart Silhouette. Hard edges are permanent edges and are present in all views. A Silhouette edge is a temporary edge that displays when the surface normal makes a 90 degree angle with the view normal. Smart Silhouettes display a silhouette only if it does not degrade the display performance.

You can set one of these as the default by choosing ***File>Preferences>Display*** and selecting the option.

Object Resolution

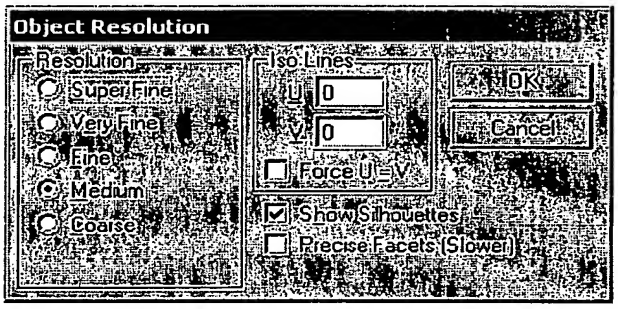
You can increase a solid's edge resolution with the Change Resolution command in the Edit menu.

You can choose one of five resolution settings, Super Fine, Very Fine, Fine, Medium or Coarse. Click a radio button next and then click OK to close the dialog box. The object will change accordingly.

Changing an Object's Resolution

1. Select the object.
2. Choose ***Edit>Change Resolution***.

   The Object Resolution dialog box appears.

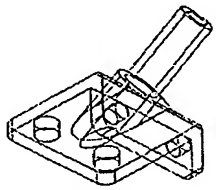


3. Enter the desired number of Iso lines. Check the Show Silhouettes box if desired.

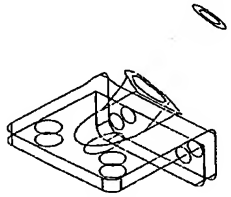
   The # Iso Lines allow you to control the isopram lines drawn for a solid face. These Iso (isopram) lines are constant parameter curves that lie on a surface, typically defined in parameter space. The parameter space coordinate system uses U and V coordinates. A 0 (zero) in both fields turns off Iso lines. The appropriate U/V values may enhance the visual appearance of the face at the expense of drawing speed. The letters, U and V are space coordinate identifiers (U = horizontal, V = vertical) that are the industry standard. The Force U=V check box automatically sets equal number of U and V lines. Entering a new value in one automatically changes the other isoline value.

   The Silhouette check box controls the silhouette edges of objects. Silhouette edges are view dependent and can cause a significant reduction in drawing speed. A check in the box turns the silhouette on.

4. Click OK to close the dialog box and save the new settings.



Silhouette On



Silhouette Off

See Chapter 24 for more information on resolution.

Solid Associativity

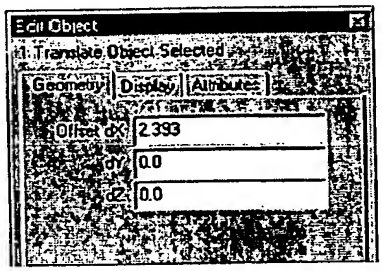
Solid objects are associative to the original curves used to create them. Changing the curve automatically changes the dependent solid. For example, if you create a lathed solid from four curves and change the length of one of them, the lathed solid automatically updates to reflect the change. Use the ***Edit>Remove Links*** command to remove all associativity with the solid if you do not want curve changes to affect the solid. Another important issue to remember with the curve/solid associativity is that if the curve is deleted, the solid deletes as well. Since it is defined by the curve, it is no longer valid.

Solids and Instances

When you perform an operation on a solid and move it to another location or copy a solid you create an instance of the original. When you display the Edit Objects dialog box, the object is referred to as an instance.

Instances and Moving Solids

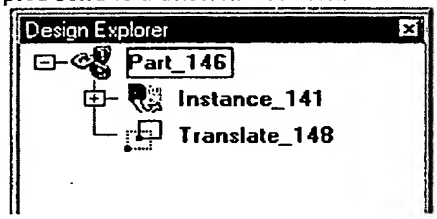
When you create an instance by moving the modified solid to another location, the information appears as an instance operation in the Design Explorer. The original object is still located in previous position, although it is not displayed.



When you select the Instance Translate item in the Design Explorer and open the Edit Objects dialog box the only geometry characteristics provided the offset distance from the original solid.

Instances, Copying Solids and Associativity

Holding down the CTRL (Windows) or OPTION (Macintosh) key while moving the object with the Selection tool creates an instance copy of the original solid. If you display the history tree in the Design Explorer, you see two instance operations. The first, called Instance, is the copying operation. The second, called Instance:Translate is the translation operation moving the copied solid to a different location.



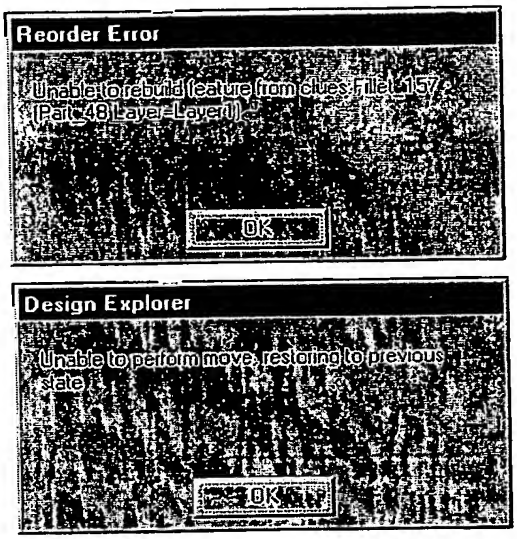
Displaying the Edit Objects dialog box for the Instance item shows there are no unique geometric characteristics. If you open the history tree further you expose the original solid and can view its characteristics through the Edit Objects dialog box.

Cobalt and Xenon Only

When you copy a solid, creating an instance, this instance is associative to the original solid. All changes made to the original solid will automatically be made on all instances created from the original. For example, if you added a hole to the original after creating the instance, the hole is also added to all instances. You may need to choose ***Edit>Resolve Links*** for this to occur.

If an operation you perform on the original after creating an instance conflicts with an operation already performed on that instance, you receive two successive errors, a Reorder

Error and a Resolve Links error (shown below). Choose **Edit>Undo** to return to part to the original state.



Cobalt and Xenon Only

When you perform operations on an instance, they are listed in the Design Explorer and do not affect the original. If you break the link between the original and the instance, changes made to the original do affect the instance.

Be aware when you are creating multiple instances whether you are instancing the original part or another instance. This is important if you later decide to break the link between the original and an instance. If there are any instances created from that instance, you are breaking their associativity to the original as well. So, if you only want to break the link between one instance and the original, all other instances should be created from the original solid.

If you do not want to create an instance, select the original solid, choose ***Edit>Copy*** and then ***Edit>Paste***. You can also select the instance and choose ***Edit>Remove Links***. However, the object geometry cannot be edited using the technique.

Cutting and Pasting Solids

When you cut and paste a solid all associativity relationships are lost. After cutting and pasting a part, you will not be able to edit any of its features. Cutting and pasting a solid

does, however, provide an easy way to remove a part's history tree, thereby minimizing the amount of memory it consumes.

Cutting and Pasting on the Macintosh

When you cut and paste a solid object into our Designer Elements program on a Macintosh computer, the object is represented in both PICT2 and VS Object form. When both forms are present, you can cut and paste a representation of the solid into other applications and cut and paste a true solid back into our Designer Elements program drawing.

Exporting Solids

If you want to export a solid model created in our Designer Elements program to another application, you must convert the model to a standard format. Our Designer Elements program supports three formats:

1. You can convert the solid into polylines or meshes with the Change Object Type command in the Edit menu. After converting the model, use the DXF export command to create an ASCII DXF file.
2. If you are exporting to an application that supports ACIS, you can generate a SAT file. SAT files contain precise geometry and topology, so the other application can import the solid's data as it was created in our Designer Elements program.
3. You can export solids through the IGES translator as solid entities or as a collection of trimmed surfaces.

Object Types and Edit Objects

Every solid object you create with our Designer Elements program tools are defined by their own characteristics and include geometric characteristics and attributes. These are displayed in the Edit Objects dialog box. While the categories listed in the Attributes tab are identical, the categories listed in the Geometry tab vary with each object.

For example: A line created by the Single Line tool includes the following categories: Length, Angle, End 1 for X, Y, and Z, and End 2 for X, Y and Z.

Chapters 18 and 19 introduce you to the solids tools of our Designer Elements program that create the objects. Included with every tool description is a list of the categories displayed for the object when Edit Objects is chosen. For information on using the Edit Objects command and the dialog box, see Chapter 24.

Error Messages

Error Messages provide more feedback when trying to perform operations such as shelling, local face modeling and so on. Our Designer Elements program tries to determine why the operation failed and provides a hint to the feature that may have caused the problem.

Solids Creation Tools - Primitives

Primitives are simple solid shapes. You can create primitives by picking one point, two points or diagonals. Each Designer Elements program primitive is defined by a unique set of characteristics which you can edit.

For each tool you can enter values in the Status Line to define a solid either before or after you create the solid. If you enter the values after you select the tool but before you create the solid, your first click in the drawing area automatically registers all Status Line values. If you enter values in the selected Status Line data field after creating the solid and while the solid is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the solid to reflect the new values.

The **Primitive** tools are explained in the order of the palette and include:

- **Sphere**
- **Slab**
- **Block**
- **Cylinder**
- **Cone**
- **Torus**
- **Prism**
- **Pyramid**

- **Ellipsoid**

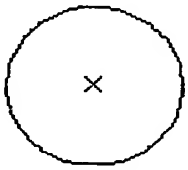
Sphere Primitive Tool

Our Designer Elements program creates a sphere from a center point and radius or from two diagonal points on a bounding box. When you select the **Sphere Primitive** tool, a subpalette appears in the Message Line containing three tools for creating spheres: **Sphere 1 Point**, **Sphere 2 Point** and **Sphere by Diagonals**.

The Drafting Assistant recognizes the points that you pick to define the sphere.

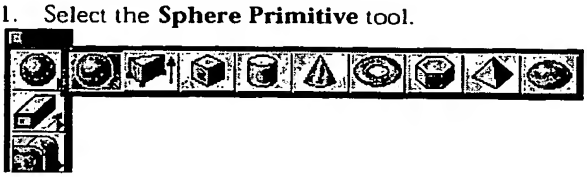
Sphere 1 Point Tool

This tool draws a sphere using the center point that you specify and the radius entered in the Status Line.



Using the Sphere 1 Point Tool

1. Select the **Sphere Primitive** tool.



2. Select the **Sphere 1 Point** tool in the Message Line. The Message Line reads: *Sphere Primitive: Enter center point for sphere.*
3. Click a point in the drawing area.

   The sphere is created in the specified location.

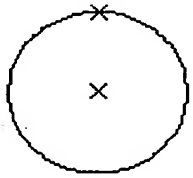
While the sphere is still selected, change the X, Y, Z location of the center point and the radius, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains X, Y and Z data fields for the sphere center point and the Radius data field.

Our Designer Elements program creates one point spheres independently of the construction plane's orientation.

Sphere 2 Point Tool

This tool draws a sphere from the center point and radius that you specify.



Using the Sphere 2 Point Tool

1. Select the **Sphere Primitive** tool.
2. Select the **Sphere 2 Point** tool in the Message Line. The Message Line reads: *Sphere Primitive: Enter center and radius point for sphere.*
3. Click the center point and the radius point in the drawing area.

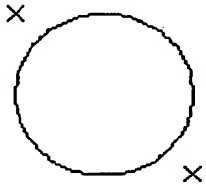
   The sphere is created.

While the sphere is still selected you can change the X, Y, Z location of the center point and dX, dY and dZ values for the radius, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains X, Y and Z data fields for the sphere center point and dX, dY and dZ fields to reference distance and direction to the radius.

Sphere by Diagonals Tool

This tool draws a sphere based on the two corners of the sphere's bounding box that you specify.



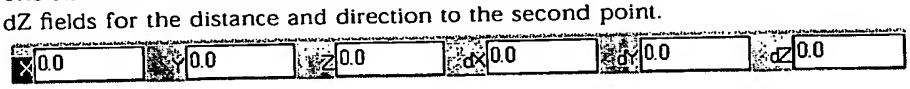
Using the Sphere By Diagonals Tool

1. Select the **Sphere Primitive** tool.
2. Select the **Sphere by Diagonals** tool in the Message Line. The Message Line reads: *Sphere Primitive: Enter diagonal corners for sphere.*
3. Click two points in the drawing area.The sphere is created.

   This Designer Elements program places the sphere's center midway between the two points and calculates the radius from the smallest length of the enclosing box.

While the sphere is still selected you can change the X, Y, Z location of the first corner point and dX, dY and dZ values to the second corner, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains X, Y and Z data fields for the first point and the dX, dY and dZ fields for the distance and direction to the second point.



Geometric Characteristics

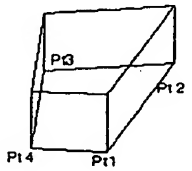
A sphere primitive is created by picking one or two points to specify the sphere's center point, radius or bounding box corners, and is made up of the following characteristics: Center (X, Y, and Z values) and Diameter. This information appears in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the sphere and choose ***Window>Edit Objects*** or double-click on the sphere.

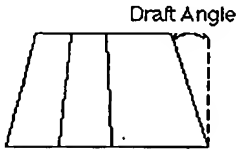
Slab Primitive Tool

You create a solid slab primitive by extruding a series of closed planar points. If the points do not close, our Designer Elements program automatically closes them. Our Designer Elements program also checks for invalid or overlapping slab points. If one or more are detected, no slab is created.

Our Designer Elements program determines the extrusion direction from the order in which you enter the points. Our Designer Elements program constructs the slab in the direction of the right hand rule (see Chapter 2 for information on the right

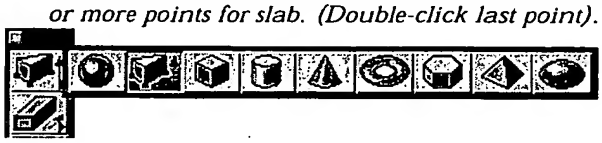
hand rule). Although the slab's points must be coplanar, they do not have to be in the current construction plane. You can also specify a draft angle for the slab.





Using the Slab Tool

1. Select the **Slab Primitive** tool. The Message Line reads: *Slab Primitive: Enter 3 or more points for slab. (Double-click last point).*



2. Click the three or more points to describe the slab. (Be careful that the points are not overlapping.)

   The slab is created with the height and draft angle listed in the Status Line.

While the slab is still selected you can change the height and draft angle in the Status Line, if desired. Press ENTER (Windows) or RETURN (Macintosh) to adjust the slab to the new values.

The Status Line contains the Height and Draft Angle data fields.

Geometric Characteristics

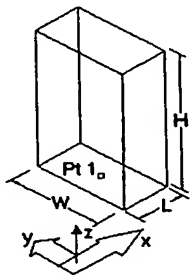
A solid slab primitive is created by picking three or more points for the slab's corners and is made up of the following characteristics: the Height and Draft Angle, and the Defining Points (X, Y, and Z values) of the slab's corner points. This information is listed in the Edit Objects dialog box under the Geometry tab. The section below Defining Points displays the selected point. Each point can be edited individually using the X, Y and Z fields. To display the dialog box, select the slab and choose ***Window>Edit Objects*** or double-click the slab.

Block Primitive Tool

The block primitive is a cubic or rectangular solid. When you select the **Block Primitive** tool a subpalette appears in the Message Line containing three block tools: **Block 1 Point**, **Block 2 Point** and **Block by Diagonals**.

Block 1 Point Tool

This tool draws a block using the center point that you specify and the Height entered in the Status Line.



Using the 1 Point Block Tool

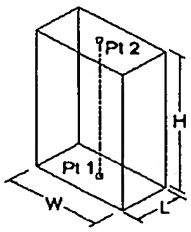
1. Select the **Block Primitive** tool.
2. Select the **Block 1 Point** tool in the Message Line. The Message Line reads: *Block Primitive: Enter center point for block.*
3. Click a point in the drawing area.

   The block is created with its base centered at the point you chose. The block's orientation is determined by its length, width, and height along the x, y, and z axes of the work plane.

While the block is still selected you can change the X, Y, Z location of the base center point and length, width and height of the block, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z location of the block base center point and Length, Width and Height of the block.

Block 2 Point Tool



This tool draws a block from the center point and radius.

Using the 2 Point Block Tool

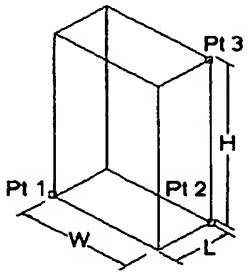
1. Select the **Block Primitive** tool.
2. Select the **Block 2 Point** tool in the Message Line. The Message Line reads: *Block Primitive: Enter center and length points for block.*
3. Click two points in the drawing area.

   The block is created centered on the point chosen with a height equal to the distance between the two points that you chose.

While the block is still selected you can change the X, Y, Z location of the base center point, the dX, dY and dZ to the second point for defining the width and height of the block and the length, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z location of the block base center point, the dX, dY and dZ to the second point for defining the width, height and length of the block.

Block by Diagonals Tool



This tool draws a block based on the two base corners and the height that you specify.

Using the Block by Diagonals Tool

1. Select the **Block Primitive** tool.
2. Select the **Block by Diagonals** tool in the Message Line. The Message Line reads: *Block Primitive: Enter start diagonal point [Shift=Square].*
3. Click two points in the drawing area to specify two diagonal corners of the block's base.

   Hold down the SHIFT key to draw a square. After you click your first point, while holding down the key, The Message Line reads: *locking....* These two points set the block's width and length.
4. After the first two points are placed The Message Line reads: *Block Primitive: Enter height.*
5. Drag to set the height of the block.

   The block is created. If you are working in the top, front or side view, pick a point, and our Designer Elements program will make the block's height equal to its width. If you are working in 3D view, you can select the point to set the desired height.

While the block is still selected you can change the X, Y, Z location of the first corner or the length, width and height of the block, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

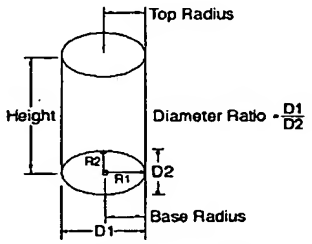
The Status Line contains the X, Y and Z values for the first corner that you chose and the solid's Length, Width and Height.

Geometric Characteristics

A solid block primitive is created by picking one or two points to specify the center point of the block's base, the block's height, or the locations of two of the block's corners. The block is made up of the following characteristics: Length, Width and Height. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the block and choose ***Window>Edit Objects*** or double-click the block.

Cylinder Primitive Tool

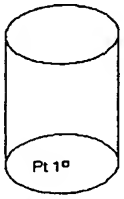
With our Designer Elements program, you can make circular and elliptical cylinder primitives. The cylinder is defined by a base radius ratio and a height. The base radius ratio defines the ratio between the major radius (R1) and minor radius (R2). A circular cylinder's base ratio equals one. Any other value yields an elliptical cylinder. The graphic gives you a visual representation of these definitions.



When you select the **Cylinder Primitive** tool, a subpalette appears in the Message Line containing three cylinder tools: **Cylinder 1 Point**, **Cylinder 2 Point** and **Cylinder by Diagonals**.

Cylinder 1 Point Tool

This tool draws a cylinder using the center point that you specify and the radius and height entered in the Status Line.



Using the Cylinder 1 Point Tool

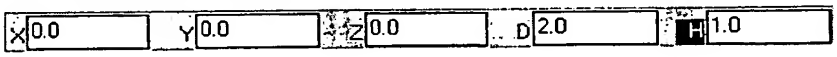
1. Select the **Cylinder Primitive** tool.

2. Select the **Cylinder 1 Point** tool in the Message Line. The Message Line reads: *Cylinder Primitive: Enter center point for cylinder.*
3. Click a point in the drawing area.

   The cylinder is created with its base centered at the point you chose. The cylinder extends along the z axis of the current construction plane.

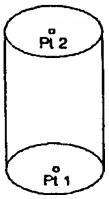
While the cylinder is still selected you can change the X, Y, Z location of the base center point and radius and height of the cylinder, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the cylinder's base center point and its Radius and Height.



Cylinder 2 Point Tool

This tool draws a cylinder from the center point of the base, the second point for the height and radius.



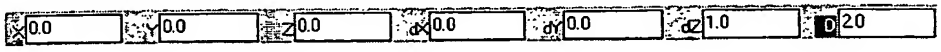
Using the Cylinder 2 Point Tool

1. Select the **Cylinder Primitive** tool.
2. Select the **Cylinder Two Point** tool in the Message Line. The Message Line reads: *Cylinder Primitive: Enter start and end points for cylinder.*
3. Click two points in the drawing area.

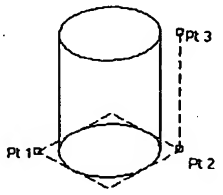
   The cylinder is created with its height is equal to the distance between the two points that you chose and with its top and bottom centered on those points.

While the cylinder is still selected you can change the X, Y, Z location of the base center point, the dX, dY and dZ to the second point for defining the distance and direction of the cylinder and the radius, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the base center point, the dX, dY and dZ values for the distance and direction to the second point, and the Radius.



Cylinder by Diagonals Tool



This tool draws a cylinder based on the two base corners and the height.

Using the Cylinder by Diagonals Tool

1. Select the **Cylinder Primitive** tool.
2. Select the **Cylinder by Diagonals** tool in the Message Line. The Message Line reads: *Cylinder Primitive: Enter start diagonal point [Shift=Square].*
3. Click two points in the drawing area to specify two diagonal corners of the cylinder base's bounding box.

   Hold down the SHIFT key to draw a circular base. After you click your first point, The Message Line reads: *locking....* The distance between the two points are used to calculate the cylinder's diameter.
4. The Message Line reads: *Cylinder Primitive: Enter height.*
5. Drag to set the height of the cylinder.

   The cylinder is created. If you are working in the top, front or side view, pick a point and our Designer Elements program will make the cylinder's height equal to that of the last cylinder you drew (or equal to one inch if this is the first cylinder you draw). If you are working in a 3D view you can select the point to set the desired height.

While the cylinder is still selected you can change the X, Y, Z location of the cylinder's base center point, the dX, dY, and dZ values for the distance and direction to the second point, and the cylinder's height, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the cylinder's base center point, the dX, dY and dZ values for the distance and direction to the second point and the Height.

Geometric Characteristics

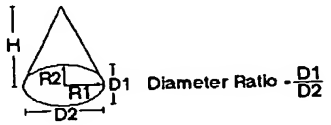
A cylinder primitive is created by picking one point to specify the center point of the cylinder's base, two points for the center point and height, or three points for

the corners of the bounding box and the cylinder's height. It is made up of the following characteristics: Height, Diameter (Major axis) and Diameter Ratio (D1/D2 or R1/R2). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the cylinder once and choose ***Window>Edit Objects*** or double-click the cylinder.

Cone Primitive Tool

The cone primitive has an elliptical base at one end and a point at the other. Like the cylinder, the cone is defined by a base radius ratio and a height.

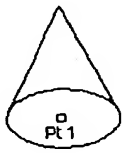
The ratio defines the relationship between the major radius (R1) and minor radius (R2). A circular cone's base ratio equals one. Any other value yields an elliptical cone.



When you select the **Cone Primitive** tool a subpalette appears in the Message Line containing three cone tools: **Cone 1 Point**, **Cone 2 Point** and **Cone by Diagonals**.

Cone 1 Point Tool

This tool draws a cone using the center point that you specify and the radius and height entered in the Status Line.



Using the Cone 1 Point Tool

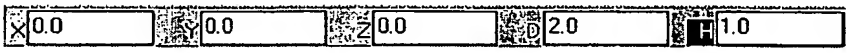
1. Select the **Cone Primitive** tool.

2. Select the **Cone 1 Point** tool in the Message Line. The Message Line reads: *Cone Primitive: Enter center point for cone.*
3. Click a point in the drawing area.

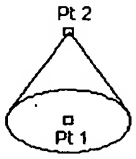
The cone is created with its base centered at the point you chose. The cone extends along the z axis of the current work plane.

While the cone is still selected you can change the X, Y, Z location of the base center point and radius and height of the cone, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the cone's base center point and its Radius and Height.



Cone 2 Point Tool



This tool draws a cone from the center point of the base at the height and radius you specify.

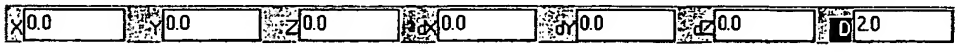
Using the Cone 2 Point Tool

1. Select the **Cone Primitive** tool.
2. Select the **Cone 2 Point** tool in the Message Line. The Message Line reads: *Cone Primitive: Enter start and end points for cone.*
3. Click in the drawing area to set the start (center) point and the end (tip) point of the cone.

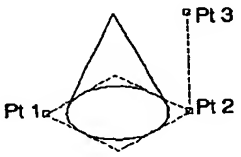
   The cone is created with its height equal to the distance between the two points that you chose and with its top and bottom centered on those points.

While the cone is still selected you can change the X, Y, Z location of the base center point, the dX, dY and dZ to the second point for defining the distance and direction of the cone and the radius, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the base center point, the dX, dY, dZ values for the distance and direction to the second point (tip) and the Radius.



Cone by Diagonals Tool



This tool draws a cone based on the two base corners and the height that you specify.

Using the Cone by Diagonals Tool

1. Select the **Cone Primitive** tool.
2. Select the **Cone by Diagonals** tool in the Message Line. The Message Line reads: *Cone Primitive: Enter start diagonal point [Shift=Square].*
3. Click two points in the drawing area to specify two diagonal corners of the cone base's bounding box.

   Hold down the SHIFT key to draw a circular base. After you click your first point, while holding down the key, The Message Line reads: *locking....*" The distance between the two points are used to calculate the cone's diameter.

   After doing this step, The Message Line reads: *Cone Primitive: Enter height.*
4. Drag to set the height of the cone.

   The cone is created. If you are working in the top, front or side view, pick a point, and our Designer Elements program will make the cone's height equal to that of the last cone you drew (or equal to one, if this is the first cone you drew). If you are working in a 3D view, you can select the point to set the desired height.

While the cone is still selected you can change the X, Y, Z location of the cone's base center point, the dX, dY, and dZ values for the distance and direction to the second point (tip), and the cone's height, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the cone's base center point, the dX, dY and dZ values for the distance and direction to the second point, and the Height.

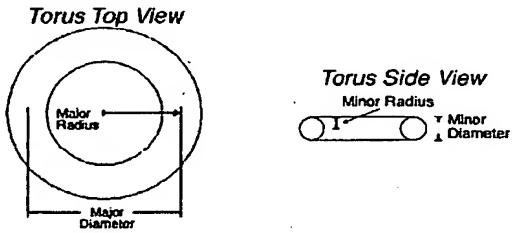
Geometric Characteristics

A cone primitive is created by picking one point to specify the center point of the cone's base, two points for the center point and height, or three points for the corners of the bounding box and the cone's height. It is made up of the following

characteristics: Height, Top Diameter, Diameter Ratio (D1/D2 or R1/R2) and Bottom Diameter. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the cone once and choose ***Window>Edit Objects*** or double-click the cone.

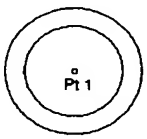
Torus Primitive Tool

The torus primitive looks like a doughnut. It is generated by sweeping a circle around another implied circle. The torus is defined by a center point, the diameter of the torus, called the Major Diameter, and the tube's diameter, called the Minor Diameter. The graphics below illustrate these dimensions.



When you select the **Torus Primitive** tool, a subpalette appears in the Message Line with three cone tools: **Torus 1 Point**, **Torus 2 Point** and **Torus by Diagonals**.

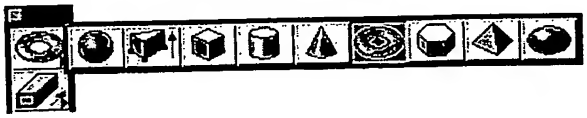
Torus 1 Point Tool



This tool draws a torus using the center point you specify and the Major Diameter (D1) and the Minor Diameter (d2) of the Status Line.

Using the Torus 1 Point Tool

1. Select the **Torus Primitive** tool.

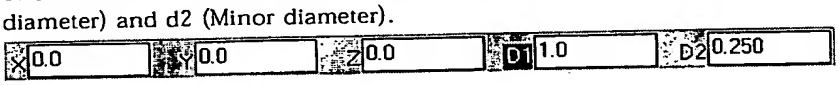


2. Select the **Torus 1 Point** tool in the Message Line. The Message Line reads: *Torus Primitive: Enter center point for torus.*
3. Click in the drawing area to set the center.

   A torus is created with its center at the point you chose. The torus lies perpendicular to the z axis of the current work plane.

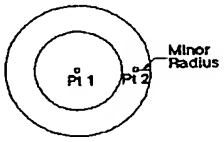
While the torus is selected you can change the X, Y, Z location of the base center point, the Major diameter (D1) and the Minor diameter (d2), if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the torus' center point, D1 (Major diameter) and d2 (Minor diameter).



Torus 2 Point Tool

This tool draws a torus from the center point and radius.



Using the 2 Point Torus Tool

1. Select the **Torus Primitive** tool.
2. Select the **Torus 2 Point** tool in the Message Line. The Message Line reads: *Torus Primitive: Enter center and radius point for torus.*
3. Click the center point and the radius point in the drawing area.

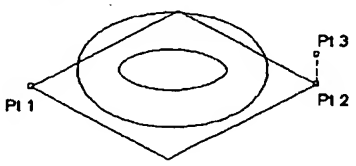
   A torus is created with a center point at the first point and the major diameter at the second point.

While the torus is still selected you can change the X, Y, Z location of the base center point, the dX, dY and dZ to the second point for defining the distance and direction, the Major Diameter (D1) and Minor Diameter (d2), if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the values.

The Status Line contains the X, Y and Z values for the torus's center point, the dX, dY and dZ values for the distance and direction to the second point, the D1 (Major Diameter) and d2 (Minor diameter).

Torus by Diagonals Tool

This tool draws a torus from the center point, major radius and minor radius.



Using the Torus by Diagonals Tool

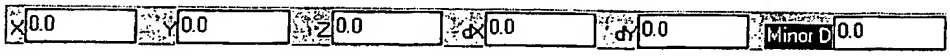
1. Select the **Torus Primitive** tool.
2. Select the **Torus by Diagonals** tool in the Message Line. The Message Line reads: *Torus Primitive: Enter start diagonal point [Shift=Square].*
3. Click twice to specify two diagonal corners of the torus' bounding box.

   Hold down the SHIFT key to draw a square bounding box. After you click your first point The Message Line reads: *locking....* The distance between the two points are used to calculate the torus' diameter.
4. The Message Line reads: *Torus Primitive: Enter height.*
5. Drag to set the height of the torus.

   If you are working in the top, front or side view, pick a point. Our Designer Elements program will calculate the torus's Minor Diameter based on the location of that point. If you are working in a 3D view, you can select the point to set the Minor Diameter.

While the torus is still selected you can change the X, Y, Z location of the torus' center point, the dX, dY, and dZ values for the distance and direction to the second point (Major Diameter), and the torus' Minor Diameter, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the torus center point, the dX, dY and dZ values for the distance and direction to the second point D1 (Major Diameter) and the d2 (Minor Diameter).

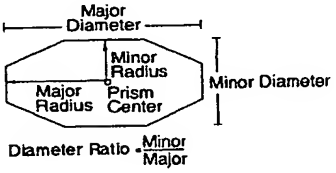


Geometric Characteristics

A torus primitive is created by picking one point to specify the center point of the torus, two points for the center point and Major Diameter, or three points for the corners of the bounding box and the torus' Minor Diameter. It is made up of the following characteristics: Major Diameter, Minor Diameter, and Center (X, Y, and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the torus and choose ***Window>Edit Objects*** or double-click the torus.

Prism Primitive Tool

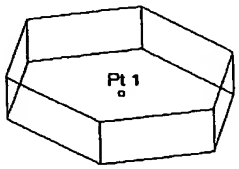
The prism primitive is composed of three or more sides, all perpendicular to the solid's base.



When you select the **Prism Primitive** tool a subpalette appears in the Message Line containing three prism tools: **Prism 1 Point**, **Prism 2 Point** and **Prism by Diagonals**.

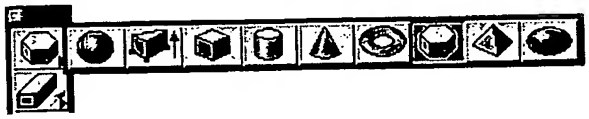
Prism 1 Point Tool

This tool draws a prism using the center point that you specify and the radius, height and number of sides listed of the Status Line.



Using the Prism 1 Point Tool

1. Select the **Prism Primitive** tool.



2. Select the **Prism 1 Point** tool in the Message Line. The Message Line reads: *Prism Primitive: Enter center point for prism.*
3. Click in the drawing area to set the base's center point.

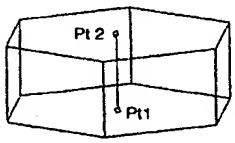
   A prism is created with its base centered at the point you chose. The prism extends along the z axis of the current work plane.

While the prism is still selected you can change the X, Y, Z location of the base center point, the radius, height and number of sides, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the prism's base center point and its Radius, Height, and # of sides.

Prism 2 Point Tool

This tool draws a prism from the center point and at the height you specify.



Using the 2 Point Prism Tool

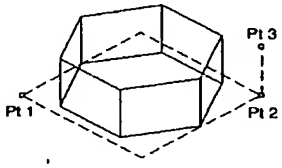
1. Select the **Prism Primitive** tool.
2. Select the **Prism 2 Point** tool in the Message Line. The Message Line reads: *Prism Primitive: Enter start and end points for prism.*
3. Click in the drawing area to set the start point (center) and the end point (height).

   A prism is created with its height equal to the distance between two points of your chosing and its top and bottom centered on those points.

While the prism is still selected you can change the X, Y, Z location of the base center point, the dX, dY and dZ to the second point for defining the distance and direction, the radius and the number of sides, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the prism's base center point, the dX, dY and dZ values for the distance and direction to the second point and the prism's Radius and # of sides.

Prism by Diagonals Tool



This tool draws a torus from two corner points, representing the prism's bounding box and a third point for the height.

Using the Prism by Diagonals Tool

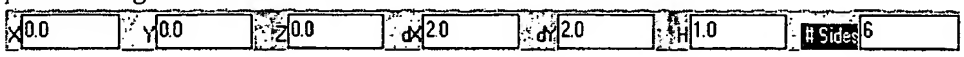
1. Select the **Prism Primitive** tool.
2. Select the **Prism by Diagonals** tool in the Message Line. The Message Line reads: *Prism Primitive: Enter start diagonal point [Shift=Square].*
3. Click two points in the drawing area to specify two diagonal corners of the prism base's bounding box.

   Hold down the SHIFT key to draw a square bounding box. After you click your first point, The Message Line reads: *locking....* The distance between the two points are used to calculate the prism's diameter.
4. The Message Line reads: *Prism Primitive: Enter height.*
5. Drag to set the height of the prism.

   If you are working in the top, front or side view, pick a point, and our Designer Elements program will make the prism's height equal to that of the last prism you drew (or equal to one inch if this is the first prism you draw). If you are working in 3D view you can select the point to set the desired height.

While the prism is still selected you can change the X, Y, Z location of the prism's base center point, the dX, dY, and dZ values for the distance and direction to the second point and the prism's height, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the prism's base center point, the dX, dY and dZ values for the distance and direction to the second point and the prism's Height.

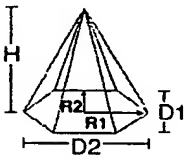


Geometric Characteristics

A prism primitive is created by picking one point to specify the center point of the prism's base, two points for the center point and height, or three points for the cor-

ners of the bounding box and the prism's height. It is made up of the following characteristics: Height, Diameter, Diameter Ratio (D2/D1 or R2/R1) and Sides. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the prism and choose ***Window>Edit Objects*** or double-click the prism.

Pyramid Primitive Tool



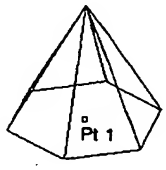
Diameter Ratio = $\frac{D1}{D2}$

A pyramid contains three or more flat faces which converge to a common vertex at the tip.

A pyramid is created by defining the diameter of the base, the height and the number of sides.

When you select the **Pyramid Primitive** tool a subpalette appears in the Message Line containing three pyramid tools: **Pyramid 1 Point**, **Pyramid 2 Point** and **Pyramid by Diagonals**.

Pyramid 1 Point Tool



This tool draws a pyramid using the center point that you specify and the radius, height and number of sides listed of the Status Line.

Using the 1 Point Pyramid Tool

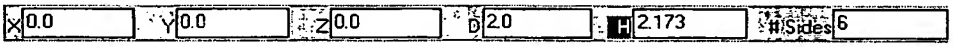
1. Select the **Pyramid Primitive** tool.

2. Select the **Pyramid 1 Point** tool in the Message Line. The Message Line reads: *Pyramid Primitive: Enter center point for pyramid.*
3. Click in the drawing area to set the center.

A pyramid is created with its base centered at the point you chose. The pyramid extends along the z axis of the current work plane.

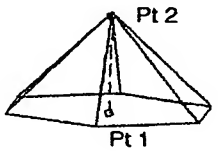
While the pyramid is still selected you can change the X, Y, Z location of the base center point, the radius, height and number of sides, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the pyramid's base center point and its radius, height and # of sides.



Pyramid 2 Point Tool

This tool draws a pyramid from the base center point and the height.



Using the 2 Point Pyramid Tool

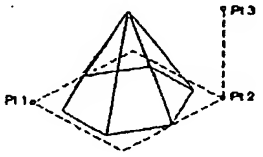
1. Select the **Pyramid Primitive** tool.
2. Select the **Pyramid Two Point** tool in the Message Line. The Message Line reads: *Pyramid Primitive: Enter start and end points for pyramid.*
3. Click two points in the drawing area.

   A pyramid is created whose height is equal to the distance between the two points that you chose and whose top and bottom are centered on those points.

While the pyramid is still selected you can change the X, Y, Z location of the base center point, the radius, height and number of sides, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the pyramid's base center point, the dX, dY and dZ characteristics for the distance and direction to the second point and the pyramid's radius and # of sides.

Pyramid by Diagonals Tool



This tool draws a pyramid from the corner of the pyramid's bounding box and the height that you specify.

Using the Pyramid By Diagonals Tool

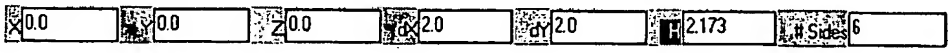
1. Select the **Pyramid Primitive** tool.
2. Select the **Pyramid by Diagonals** tool in the Message Line. The Message Line reads: *Pyramid Primitive: Enter start diagonal point [Shift=Square].*
3. Click two points in the drawing area to specify two diagonal corners of the pyramid base's bounding box.

   Hold down the SHIFT key to draw a square bounding box. After you click your first point, The Message Line reads: *locking....* The distance between the two points are used to calculate the pyramid's diameter.
4. The Message Line reads: *Pyramid Primitive: Enter height.*
5. Drag to set the height of the pyramid.

   If you are working in the top, front or side view, pick a point, and a pyramid is created with a height equal to that of the last pyramid you drew (or equal to one inch if this is the first pyramid you draw). If you are working in a 3D view you can select the point to set the desired height.

While the pyramid is still selected you can change the X, Y, Z location of the pyramid's base center point, the dX, dY, and dZ values for the distance and direction to the second point and the pyramid's height, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the pyramid's base center point, the dX, dY and dZ values for the distance and direction to the second point and the pyramid's Height.



Geometric Characteristics

A pyramid primitive is created by picking one point to specify the center point of the pyramid's base, two points for the center point and height, or three points for the corners of the bounding box and the pyramid's height. It is made up of the following characteristics: Height, Top Diameter, Diameter Ratio (D2/D1 or R2/R1),

Bottom Diameter and Sides. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the pyramid and choose ***Window>Edit Objects*** or double-click the pyramid.

Ellipsoid Primitive Tool

An ellipsoid primitive is created from a center point, center point and radius, or two diagonal points on a bounding box.

When you select the **Ellipsoid Primitive** tool a subpalette appears in the Message Line containing three tools for creating ellipsoids: **Ellipsoid 1 Point**, **Ellipsoid 2 Point** and **Ellipsoid by Diagonals**.

Ellipsoid 1 Point Tool

This tool creates an ellipsoid using the center point that you specify and the X, Y and Z diameters entered in the Status Line. The Drafting Assistant displays a *vertex* notation at the center point.

Using the Ellipsoid 1 Point Tool

1. Select the **Ellipsoid Primitive** tool.

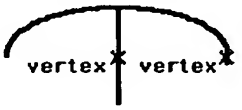
2. Select the **Ellipsoid 1 Point** tool in the Message Line. The Message Line reads: *Ellipsoid Primitive: Enter center point for ellipsoid.*
3. Click a point in the drawing area.

   The ellipsoid appears.

While the ellipsoid is still selected you can change the X, Y and Z location of the center point and the diameters. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the center point and DiamX, DiamY and DiamZ for the ellipsoid diameters.

Ellipsoid 2 Point Tool



This tool creates an ellipsoid using the center point and the radius you specify. The Drafting Assistant displays *vertex* notations at the center point and radius.

Using the Ellipsoid 2 Point Tool

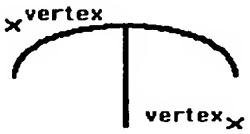
1. Select the **Ellipsoid Primitive** tool.
2. Select the **Ellipsoid 2 Point** tool in the Message Line. The Message Line reads: *Ellipsoid Primitive: Enter center and radius point for ellipsoid.*
3. Click the center point and radius point in the drawing area.

   The ellipsoid appears.

While the ellipsoid is still selected you can change the X, Y and Z location of the center point, the dX, dY and dZ values for the first radius and the two radii not specified in the operation (R1 and R2). Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

e Status Line contains the X, Y and Z values for the center point, the dX, dY, dZ, R1 and R2 values.

Ellipsoid by Diagonals Tool



This tool creates an ellipsoid based on two corners of the ellipsoid's bounding box and the height you specify. The Drafting Assistant displays *vertex* notations at the bounding box corners.

Using the Ellipsoid by Diagonals Tool

1. Select the **Ellipsoid Primitive** tool.

2. Select the **Ellipsoid by Diagonals** tool in the Message Line. The Message Line reads: *Ellipsoid Primitive: Enter diagonal corners for ellipsoid. Enter start diagonal point [Shift = Square].*
3. Click two points in the drawing area to specify two diagonal corners of the ellipsoid's bounding box.

   Hold down the SHIFT key to draw a rectangular base. After you click your first point The Message Line reads: *locking...* The distance between the two points are used to calculate the ellipsoid's diameter.
4. The Message Line reads: *Ellipsoid Primitive: Enter height.*
5. Drag to set the height of the ellipsoid.

   The ellipsoid appears. If you are working in the top, front or side view, pick a point, and our Designer Elements program will make the ellipsoid's height equal to that of the last ellipsoid you drew (or equal to one inch if this is the first ellipsoid you draw). If you are working in a 3D view you can select the point to set the desired height.

While the ellipsoid is still selected you can change the X, Y, Z location of the ellipsoid's base center point, the dX, dY, and dZ values for the distance and direction to the second point, and the ellipsoid's height. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the cylinder's base center point, the dX, dY and dZ values for the distance and direction to the second point.

Geometric Characteristics

An ellipsoid primitive is created by picking one point to specify the center of the primitive's base, two points for the center point and height, or three points for the corners of the bounding box and the ellipsoid's height. It is made up of the following characteristics: Center X, Y and Z and Diameter 1, Diameter 2 and Diameter 3.

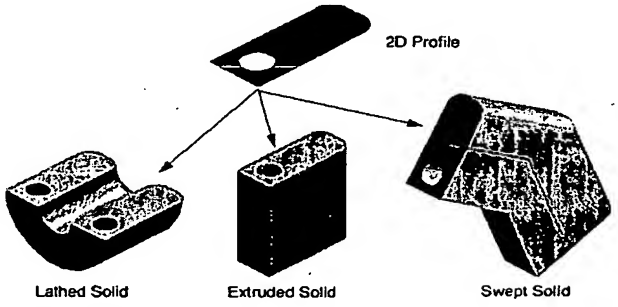
For ellipsoids created using the **Ellipsoid 1 Point** tool, DiamX, DiamY and DiamZ correspond to Diameter1, Diameter2 and Diameter3. For ellipsoids created with the **Ellipsoid 2 Point** tool, the dX, dY, dZ values are used to determine the Diameter 1 value. R1 corresponds to Diameter 2 and R2 corresponds to Diameter 3.

This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the cylinder and choose ***Window>Edit Objects*** or double-click on the cylinder.

Solids Creation Tools - Profiles

A profile is a closed section created from curves, polygons or surfaces (planar and non-planar). You can lathe, extrude, or sweep a profile to create a solid.

When you make a solid from a profile, our Designer Elements program creates a parent/child relationship **(Cobalt™ and Xenon™ only)** between the profile (the parent) and the solid (the child). If you change the parent, our Designer Elements program automatically regenerates the child **(Cobalt™ and Xenon™ only)**. For example, if you lathe a rounded rectangle polygon, you can change the polygon's radius and the solid will update to the new size **(Cobalt™ and Xenon™ only)**. The graphic here shows how you can use the same 2D profile to create three different objects.



Our Designer Elements program supports five tools for creating solids from profiles. For each tool you can enter values in the Status Line to define a solid, either before or after you create the solid. If you enter the values after you select the tool but before you create the solid, your first click in the drawing area automatically registers all Status Line values. If you enter values in the selected Status Line data field

Tech Note:
The ASIC kernal has been improved such that you may now use grouped profiles as well as grouped curves for profiles..

Tech Note:
These tools do not support using the *Select Chain* command to select multiple grouped profiles. However can can extrude multiple grouped profiles at one time.

after creating the solid and while the solid is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the solid to reflect the new values.

The topics explained include the following:

- **Lathed Solid** tool
- **Extrude Solid** tool
- **Swept Solid** tool
- **Swept Solid 2 Rail** tool
- **Cutout Feature** tool
- **Protruded Feature** tool
- **Skinned Solid** tool
- **Pipe Solid** tool
- Solids from Profile Curve Checks

Lathed Solid Tool

The Lathed Profile tool creates a solid by revolving a profile around an axis line. You can lathe profiles composed of individual, grouped or nested group curves. You cannot lathe a profile that intersects itself, as can happen with splines and polygons.

Using the Lathed Solid Tool

Before you can use this tool you must first create a profile and a curve for lathe axis.

1. Click the Lathe Solid tool. *The Message Line reads: Lathed Solid: Select objects to lathe [Shift=Extend].*
2. The Status Line contains the Degree and Draft Angle fields. Enter the desired values for your lathed solid. Tab between data fields.

When the draft angle is zero, the sides of the solid lie parallel to the profile normal. If the draft angle is positive, the sides will be forced outward from the nor-

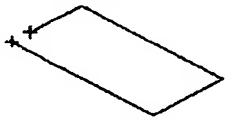
mal. Negative draft angles force the sides inward towards the normal.

3. Pick a curve or curves to lathe. If you are selecting more than one curve, hold down the SHIFT key before selecting the first curve.

   The curves making up the profile must be closed. If they are not, this message appears:

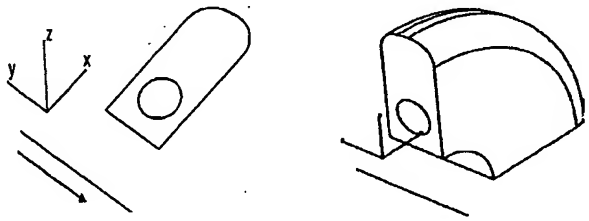
   Our Designer Elements program places markers at the gap locations so you can easily find and close them.

   The Message Line now reads, Lathed Solid: Pick line for lathe axis [Shift=Extend].

4. Pick the line around which to rotate the curves and make the solid object.

   Our Designer Elements program determines the direction of revolution from the direction in which the line was drawn. The left graphic shows a view of a profile and a line drawn in the direction of the arrow. The right graphic shows the profile lathed 90°.

   After you click the line, the object is lathed.

While the object is still selected you can edit the number of degrees in the revolution angle and the draft angle for the entire solid. Enter the new values and press ENTER (Windows) or RETURN (Macintosh) and the solid updates.

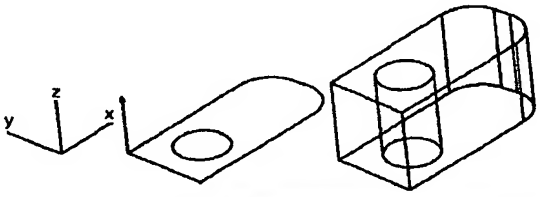
Geometric Characteristics

A lathed profile solid is created by picking one or more curves from which to construct the solid and a line about which to rotate the curves. The lathed solids is

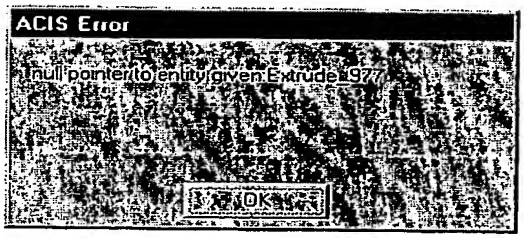
made up of the following characteristics: Lathe Angle and Draft Angle. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the lathed solid and choose **Window>Edit Objects** or double-click the lathed solid.

Extrude Solid Tool

The Extrude Solid tool extends the selected profile along a vector. You can extrude profiles composed of lines, circles, ellipses, closed splines, polygons, grouped curves (including nested groups) and surfaces.



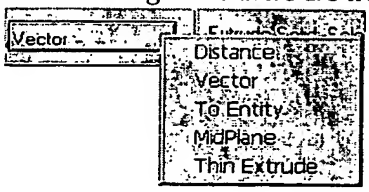
Be careful when selecting circles for extrusion. If one of them is actually a sphere you will receive the following error message:



This tool does not support extruding a non-planar surface with a non-zero draft angle. This message displays if you attempt to do so.

In our Designer Elements program, this tool uses the plane of the selected profile for the extrusion rather than the work plane normal, providing greater flexibility.

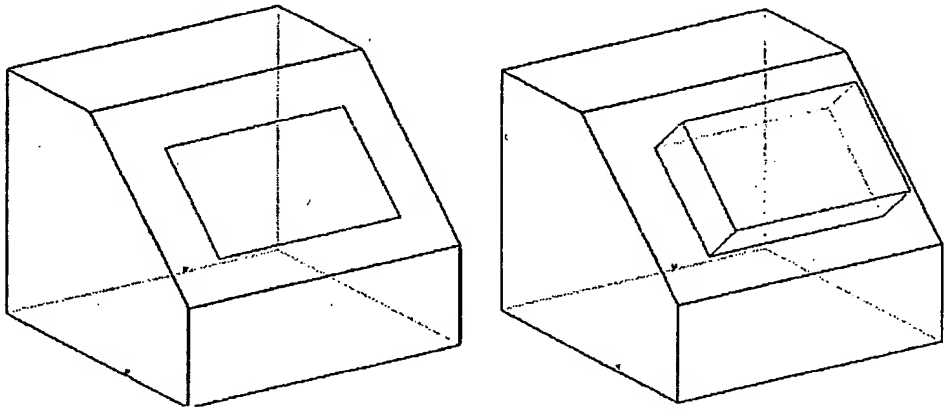
In the Message Line there are five extrusion options:



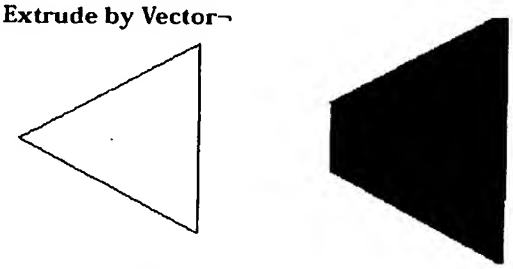
Distance	Extrudes normal to the plane of the profile a specified distance. The distance can be positive or negative.
Vector	Extrudes a profile along a vector whose distance is defined by the user supplied points.
Mid Plane	Extrudes a profile in both directions using the plane of the profile.
To Entity	Extrudes along a vector terminating at a surface or solid. The terminating body is not unioned with the extruded body.
Thin	Extrudes an open or closed profile by a vector and thickness.

Below are examples of each type of extrusion

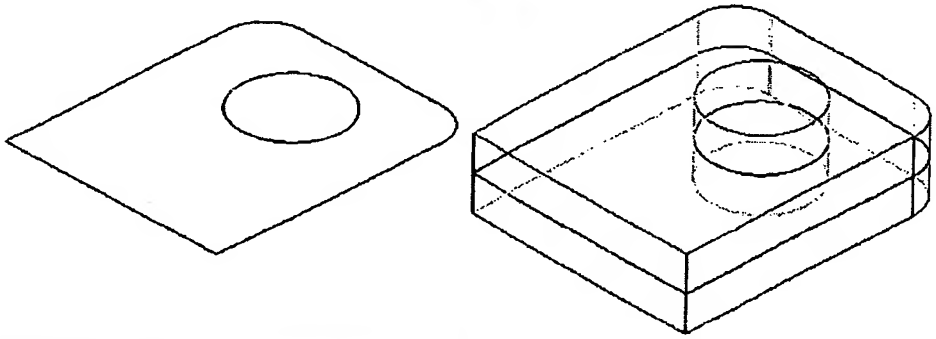
Extrude by Distance



Extrude by Vector¬

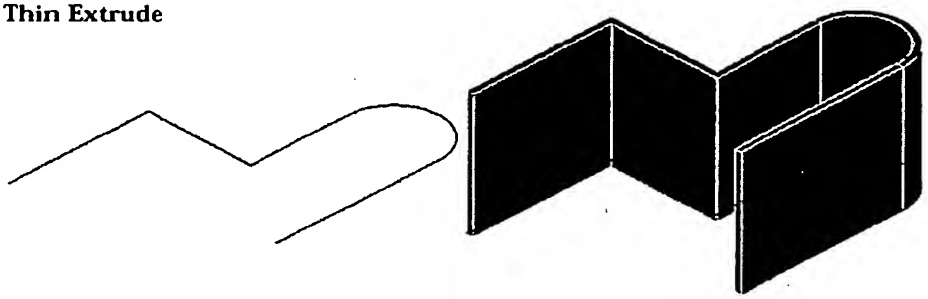


Extrude by Mid Plane

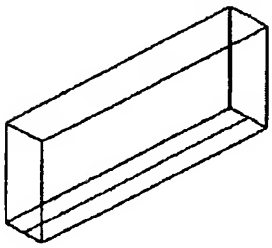


Extrude to Entity

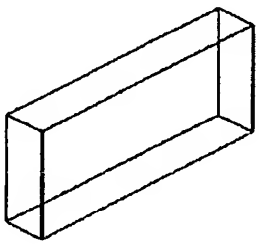
Thin Extrude



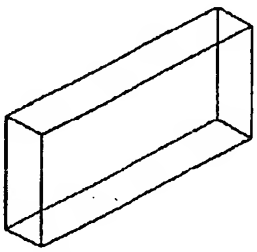
Note: The Thin Extrude tool has a toggle key that changes the location of the extrude. Use the CRTL key (Windows) or the OPTION key (Macintosh) to toggle the extrusion between the right side, left side or offset (mid plane) directions. Examples of the toggle directions are below.



Offset/Mid Plane

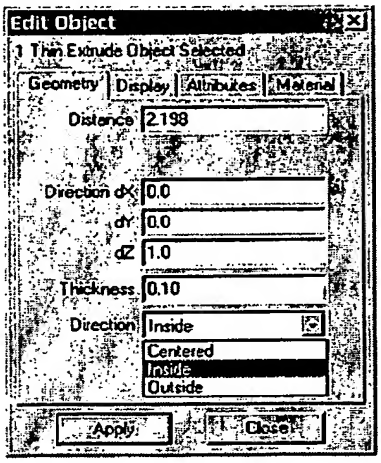


Right Side Extrude



Left Side Extrude

You have access to these toggle directions in the Edit Objects dialog box. However you must access the Edit Objects box through the Design Explorer in order to change these directions.



Using the Extrude Solid Tool

Before you can use this tool you must first create a profile to extrude.

1. Click the Extrude Profile tool. *The Message Line reads: Extrude Solid: Select close objects to extrude by vector [Shift=Extend].*
2. The Status Line contains the dX, dY and dZ values, the Distance of the extrusion vector and the Draft Angle. Enter the desired Distance and the Draft Angle for your extruded solid. Tab between data fields.

When the draft angle is zero, the sides of the solid lie parallel to the profile normal. If the draft angle is positive, the sides will be forced outward from the normal. Negative draft angles force the sides inward towards the normal.

Tech Note:

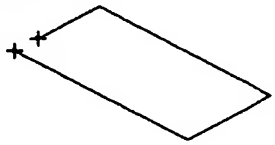
Some draft angles and extrusion heights will cause the solid to self intersect. If this happens you must reduce the draft angle or increase the extrusion height.

If you select multiple profiles and one of them is not planar, a warning will appear. You must correct this or you will receive an ACIS error.

3. Pick one or more curves to extrude. As the Message Line indicates, they must be closed. If they are not, this message appears:

Our Designer Elements program places markers at the gap locations so you can easily find and close them.

The Message Line now reads, Extrude Solid: Specify two points for extrusion direction and length.



4. Pick two points on the screen. The extruded solid is created.

If the extrusion vector defined by is points travels in a positive direction, the extrusion lies along the profile's normal. If the extrusion vector is negative, the extrusion direction lies opposite the profile's normal.

The following error message appears if you choose points that are tangential to the profile:

While the object is still selected you can edit the dX, dY, and dZ values for the extrusion vector's of the extrusion vector's distance and the draft angle for the entire solid. Enter the new values and press ENTER (Windows) or RETURN (Macintosh) and the solid updates.

Tech Note:

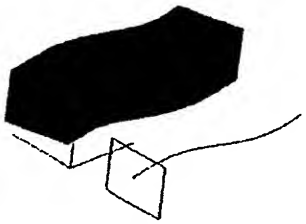
You must enter the extrusion distance for each new profile recreated even if the distance is the same as the previous extrusion.

Geometric Characteristics

An extruded profile solid is created by picking one or more curves from which to construct the solid, and then picking two points to indicate the extrusion's distance and direction. It is made up of the following characteristics: Distance, Draft Angle, and the Direction of the vector (DX, DY and DZ). This information is listed in the

Edit Objects dialog box under the Geometry tab. To display the dialog box, select the extruded solid and choose **Window>Edit Objects** or double-click the extruded solid.

Swept Solid Tool



The Swept Solid tool creates a solid by sweeping the profile along a curve. You can extrude profiles composed of lines circles, ellipses, closed splines, polygons, grouped (including nested groups) curves and surfaces.

Important: Be careful that there are no gaps or overlapping curves in the profile.

The Swept Solid tool has two pull down options in the Message Line. The first pull down controls how the profile orientation as it is swept along the path. This pull down has three options:

- **Sweep In Place** - The profile is not translated or aligned to the path.
- **Sweep Perp** - The profile is translated and aligned perpendicular to the path
- **Sweep Rigid** - The profile orientation is maintained regardless of path tangency.

The second pull down controls how the profile is terminated. This pull down has three options:

- **Curve Extents** - The profile is swept across the entire curve extents.
- **To Body** - The profile is terminated at a surface or solid.
- **Between Points** - The user specifies two points on the path.

Note: Sweep Perpendicular was the default sweep behaviour in previous version of Cobalt and Vellum Solids.

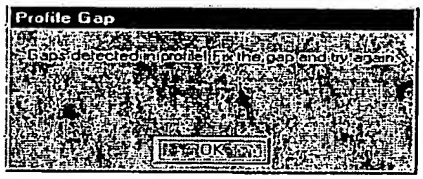
Using the Swept Solid Tool

Before you use this tool you must first create a profile and a curve for the sweep.

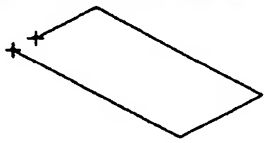
1. Click the Swept Solid tool. The Message Line reads: *Swept Solid: Select closed profile to sweep [Shift=Extend].*

2. The Status Line contains the Twist Angle and Draft Angle data fields. Enter the twist angle through which you want your solid to twist from beginning to end and a draft angle if desired.

3. Pick one or more curves for the profile. As the Message Line indicates, they must be closed. If they are not, this message appears:



Our Designer Elements program places markers at the gap locations so you can easily find and close them.



The Message Line now reads, *Swept Solid: Enter location for sweep profile origin.*

4. Pick the point on the screen where you want the sweep to begin.

The Message Line now reads, *Swept Solid: Pick curve(s) for sweep path [Shift=Extend].*

5. Pick the curve or curves along which to sweep the profile.

The swept solid is created.

While the object is still selected you can edit the twist and draft angles. Enter the new values and press ENTER (Windows) or RETURN (Macintosh) and the solid is adjusted.

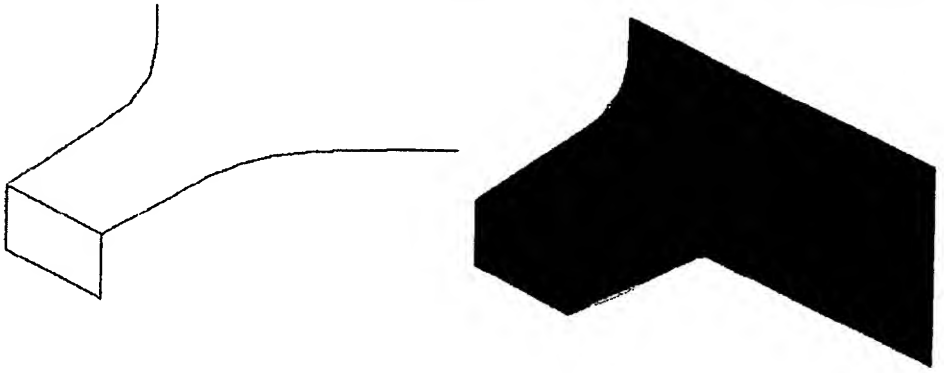
Geometric Characteristics

A swept profile solid is created by picking one or more curves from which to construct the solid, picking the point where the sweep will start, and then picking the curve along which the profile will sweep. It has two characteristics, Twist Angle and Draft Angle. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the swept solid once and choose **Window>Edit Objects** or double-click the swept solid.

Sweep 2 Rail (Paths) Solid

The Sweep 2 Rail Solid tool creates a solid by sweeping the profile along two curve. You can extrude profiles composed of lines circles, ellipses, closed splines, polygons, grouped curves (including nested groups) and surfaces.

Important: Be careful that there are no gaps or overlapping curves in the profile.



Using the Sweep 2 Rail Solid Tool

Before you use this tool you must first create a profile and a curve for the sweep.

1. Select the Sweep 2 Rail Solid tool. The Message Line reads: *Sweep 2 Rail Solid: Pick curves or group to sweep. [Ctrl = Maintain Height]/[Shift = Extend]*
2. Pick the curves or group that defines the profile you want to sweep.
3. Choose the first rail that your profile will be swept along.
4. Choose the second rail that your profile will be swept along.

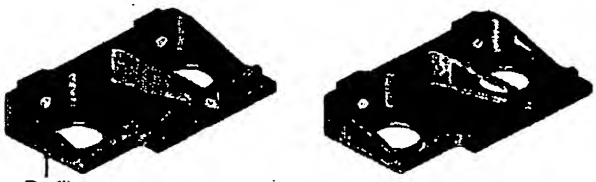
There are not status line entries for this tool.

Geometric Characteristics

A swept 2 rail profile solid is created by picking closed curves or a group from which to construct a solid. Picking the two rails (curves) along which the profile will be swept. There are only to tabs in the Edit Objects dialog box for this tool, Display and Attributes.

Cutout Feature Tool

This tool subtracts material from a solid. The cutout is formed from a 2D profile. This profile can be located on or in the solid and composed of grouped curves. You can specify a Draft Angle in the Status Line.



Using the Cutout Feature Tool

1. Create a 2D profile with a curve or polygon tool and place it on or in the solid where the cutout will begin.
2. Select the Cutout Feature tool. The Message Line reads: *Cutout Feature: Pick solid for cutout.*

3. Select the solid to be cut. The Message Line now reads, *Cutout Feature: Pick closed curves or polygon for cutting [Shift=Extend].*
4. Select the profile previously created.
5. Specify two points in the drawing area to indicate the direction and length of the cutout.

While the solid is still selected you can change the distance, draft angle and direction of the cutout in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

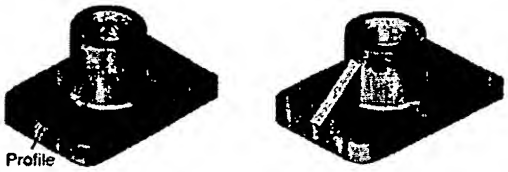
The Status Line contains the extrusion dX, dY and dZ values, the extrusion Distance and the Draft Angle.

Geometric Characteristics

A cutout is created by selecting a profile, the solid through which to cut and the distance and direction of the cut. It is made up of these characteristics according to Edit Objects: Distance, Draft Angle and Direction (DX, DY, DZ values). To modify the hole characteristics, select the hole using the Design Explorer and choose **Window>Edit Objects** or double-click on the cutout name. See Chapter 24 for more information on the Design Explorer.

Protruded Feature Tool

The tool adds a piece to an existing solid. The protrusion is formed from a 2D profile. The profile does not need to be attached to the original solid and can be grouped curves.



Termination Types

The Message Line also contains a pull-down menu allowing you to choose where the protrusion will end. You have two options: Vector and To Face.

Vector — Defined by two points that you choose the vector determines the length and direction of the protrusion.

To Face — Extends the protrusion to a face on the solid.

Using the Protruded Feature Tool - Vector Option

1. Create a 2D profile where the protrusion will begin.
2. Select the Protruded Feature tool. The Message Line reads: Protruded Feature: Pick solid for protrusion.

3. Select the Vector Termination Type from the pull-down menu in the Message Line.
4. Select a solid. The Message Line reads: *Protruded Feature: Pick closed curves or polygon for adding [Shift=Extend].*
5. Select the profile previously created.
6. Specify two points for protrusion direction and length.

While the solid is still selected, you can change the direction and distance of the protrusion and the draft angle in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

Using the Protruded Feature Tool - Face Option

1. Create a 2D profile where the protrusion will begin. (It need not be on the solid.)
2. Select the Protruded Feature tool. The Message Line reads: *Protruded Feature: Pick solid for protrusion.*
3. Select the To Face Termination Type from the pull-down menu in the Message Line.
4. Select a solid. The Message Line reads: *Protruded Feature: Pick closed curves or polygon for adding [Shift=Extend].*
5. Select the profile previously created.
6. Select the face on the solid where you want the protrusion to end.
7. Specify two points for the protrusion direction. Since you are specifying the direction, not the length, you do not need to reference the curve or the solid.

While the solid is still selected you can change the direction and distance of the protrusion and the draft angle in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the protrusion dX, dY and dZ values, the protrusion's Distance and Draft Angle.

Geometric Characteristics

A protrusion is created by selecting a profile, the solid from which you want to extend the profile and the distance and direction of the protrusion. It is made up of

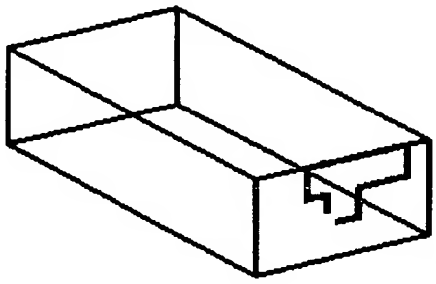
these characteristics according to Edit Objects: Distance, Draft Angle and Direction (DX, DY, DZ values). It also contains the Offset Both Directions check box. To modify the protrusion characteristics, select the protrusion using the Design Explorer and choose **Window>Edit Objects** or double-click on the protrusion name. See Chapter 24 for more information on the Design Explorer.

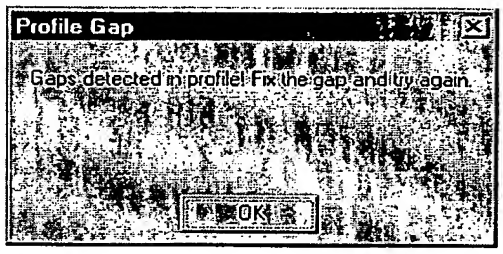
Cutout and Protruded Curve Checks

To assist you in performing cutout and protrude operations, this Designer Elements program provides you with two curve checks: **gaps** and **planar**.

Curve Gap Check

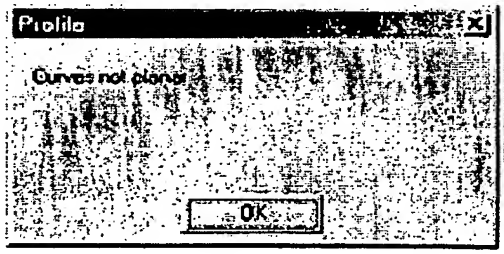
The gap check warns you when the curves for the cutout or protrusion are not closed.

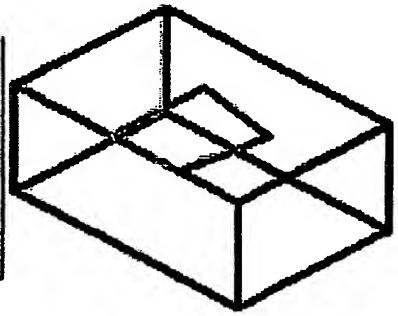




Planar Curves Check

The planar check warns you if the selected curves for the cutout or protrusion are not planar.





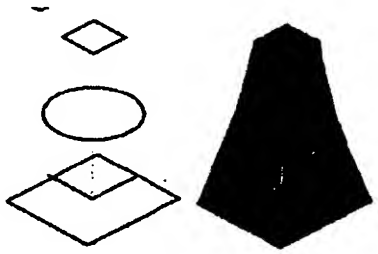
Skinned Solid Tool

The Skinned Solid tool creates a solid from a collection of closed profiles. You can create skin solids using profiles composed of individual curves (such as circles or ellipses) or grouped curves. A profile composed of individual curves does not have to be grouped for use with this tool.

There are two tools in the Message Line: Skin Curve and Skin with Guide Curves.

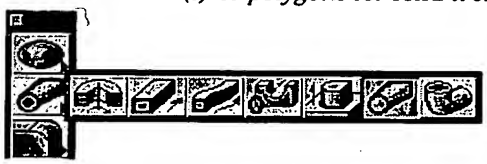
Skin Curve Solid Tool

This tool creates a skin solid between two or more curves. The solid edges are defined by the limits of the select curves. The figure here shows a skin solid created from three curves.



Using the Skin Curve Solid Tool

1. Select the Skinned Solid tool. The Message Line reads: *Skinned Solid: Pick closed curve(s) or polygons for solid from sections [Shift = Extend].*



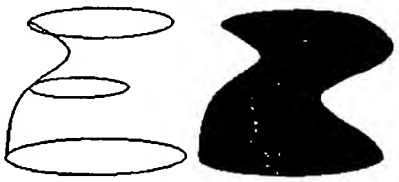
2. Select the Skin Curve Solid tool in the Message Line.
3. Hold down the SHIFT key and select the closed curves.

   The solid is created.

There are no entries in the Status Line.

Skin with Guide Curves Solid Tool

This tool creates a skinned solid between two or more curves using curves as guides to define the edges. Guide curves give you control over the skinned solid. You can use one or more guide curves to influence the solid. In the graphic the circles are skinned using the spline as a guide.



Guide Curve Rules

- Curves can go in any direction and need not be consistent with the others.
- Curves cannot loop and must be "well-behaved."
- The curves must connect with each profile that you want to use for the solid creation.

Using the Skin with Guide Curves Solid Tool

1. Create the profile curves and curve guides.
2. Select the Skinned Solid tool. The Message Line reads: *Skinned Solid: Pick closed curve(s) or polygons for solid from sections [Shift = Extend].*
3. Select the Skin with Guide Curves Solid tool (the right tool) in the Message Line.
4. Hold down the SHIFT key and select the closed curves.

   The Message Line now reads, *Skinned Solid: Pick guide path for skinned solid [Shift = Extend].*
5. Select the guide curves.

   The solid is created.

There are no entries in the Status Line.

Geometric Characteristics

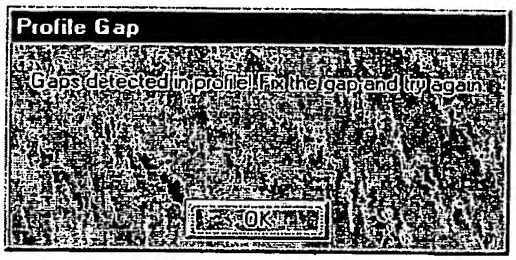
A skinned solid object is a solid created from profiles and is made up of the same characteristics as any solid object.

Solids from Profiles Curve Checks

In order to create objects using the Lathed Solid, Extrude Solid and Swept Solid tools, the profiles must be closed and planar.

Close Curves Check

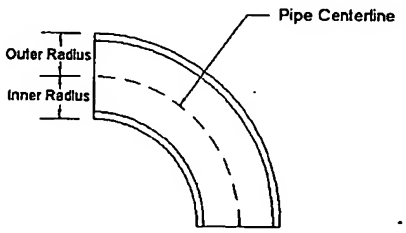
The closed curves check warns you if your profile contains gaps.



Pipe Solid Tool

The Pipe Solid tool provides a quick and easy way to create pipes and tubes.You select a curve for the pipe's center line and specify the pipe's inner and outer diameter values. You can create a pipe by selecting curves connected end to end. You cannot create more than one pipe at a time with this tool.

When you make a pipe solid, this Designer Elements program creates an intelligent link between the solid and its parent center line. When you move or modify the center line, the pipe updates automatically. A pipe with an inner diameter of zero is a solid cylinder.



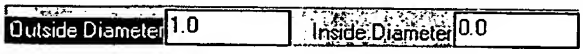
This tool does not support grouped curves for creating a pipe solid.

Using the Pipe Solid Tool

Before you can use this tool you must first create a curve for the pipe.

1. Click the Pipe Solid tool. The Message Line reads: *Pipe Solid: Select curve for pipe path [Shift=Extend].*

   The Status Line contains the Outside Diameter and Inside Diameter data fields.



2. Enter values in the Status Line data fields.

   An Inside Diameter of zero creates a cylinder. Tab between the fields.

3. Select the curve. If you want to select more than one curve, hold down the SHIFT key before selecting the first curve.

   A pipe is drawn.

While the object is still selected you can edit the Outside Diameter and the Inside Diameter. Enter the new values and press ENTER (Windows) or RETURN (Macintosh) and the solid is adjusted.

Geometric Characteristics

A pipe solid is created by selecting one or more curves and entering specifying the inner and outer diameters. A pipe solid is made up of the following characteristics: Outer Diameter, Inner Diameter and Wall Thickness. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the extruded solid and choose **Window>Edit Objects** or double-click the pipe.

General Editing Tools

Once you create an object you can select and alter it by choosing a command or using a tool to edit it. This chapter describes common editing activities and compares different methods for performing the same action. You can use the editing tools to change an object's geometry, size, location or orientation. They are collected in three subpalettes in the main tool palette.

For each tool you can enter values in the Status Line to perform the editing operation before or after the operation is complete. If you enter the values after you select the tool but before you perform the operation, your first click in the drawing area automatically registers all Status Line values. If you enter values after performing the operation and while the curve is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the curve to reflect the new values.

Tech Note:

These editing tools cannot be performed on grouped objects, only curves.

Many of the editing tools require a two step process for their usage. The first step after choosing the tool may be to select one or more objects. The second step performs the editing operation. Once you the complete the first step you cannot select more objects to include in the same operation. To select additional objects reset the tool and start again.

There are no unique geometric characteristics listed in the Edit Objects dialog box for an object after using these tools. For example, a fillet between two lines creates an arc. Edit Objects lists the parameters for an arc.

The following topics are explained:

- **Fillet/Chamfer** tools
- **Trim** tools

- **Curve Extras** tools

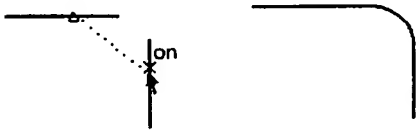
Fillet and Chamfer Tools

These tools on the main tool palette construct fillets and chamfers from corners formed by nonparallel lines or curves. This Designer Elements program automatically trims fillets and chamfers by default. If you do not want them trimmed hold down the CTRL (Windows) or the OPTION (Macintosh) key while you are selecting the objects to fillet or chamfer.

All curves that you want to fillet must be in the same plane.

2-Entity Fillet Tool

This tool constructs an arc tangent to the two objects you specify.

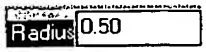


Using the 2-Entity Fillet Tool

1. Select the **2-Entity Fillet** tool. The Message Line reads: *2-Entity Fillet: Pick first entity. [Shift=Corner, Ctrl=No trim* (Windows) or *Option=No Trim (Macintosh)].*
2. Enter the arc radius you want in the Status Line. The default radius is .50 inch.
3. Click the first curve. The Message Line reads: *2-Entity Fillet: Pick second entity. [Ctrl=No trim]* (Windows) or *[Option=No Trim* (Macintosh)*].*

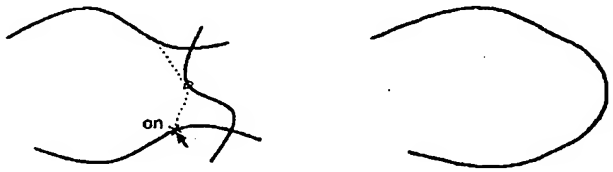
You can also hold down the SHIFT key and click once inside the corner you want to fillet. (See the Message Line at the top of your drawing area for a reference to this feature.) If you hold down the CTRL (Windows) or OPTION (Macintosh) key while you select the objects to fillet the objects are not trimmed. (See the Message Line). Fillets create the smallest possible arc between the selected geometry.

The Status Line contains the Radius of the fillet.



3-Entity Fillet Tool

This tool constructs a fillet tangent to the three objects you choose.



Using the 3-Entity Fillet Tool

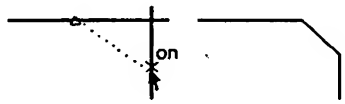
1. Select the **3-Entity Fillet** tool. The Message Line reads: *3-Entity Fillet: Pick three entities [Ctrl=No Trim* (Windows) or *Option=No Trim* (Macintosh)*].*

2. Click the three objects you want to fillet.

If you hold down the CTRL (Windows) or OPTION (Macintosh) key while you select the objects to fillet the objects are not trimmed (see the Message Line).

There are no Status Line entries.

2-Entity Chamfer Tool



This tool creates a chamfer across a corner at the specified distance from the intersection of two lines.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when creating a chamfer with this tool.

Using the 2-Entity Chamfer Tool

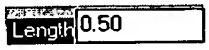
1. Select the **2-Entity Chamfer** tool.

2. In the Status Line specify the chamfer's distance from the corner. (The default distance is .50 inch.)
3. Click the first curve. The Message Line reads: *2-Entity Chamfer: Pick second entity. [Ctrl=No trim* (Windows) or *Option=No Trim* (Macintosh)*].*

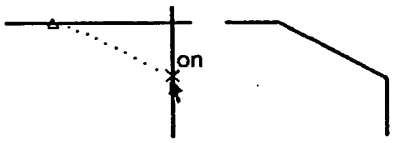
You can also hold down the SHIFT key and click once inside the corner you want to chamfer. (See the Message Line at the top of your drawing area for a reference to this feature.)

The lines are automatically trimmed or extended. If you hold down the CTRL (Windows) or the OPTION (Macintosh) key while you select the objects to chamfer, the objects are not trimmed (see the Message Line).

The Status Line allows you to specify the distance (Length) from the chamfer to the intersection of the corner lines.



Angular Chamfer Tool



This tool creates a chamfer at the specified angle and distance from the corner.
The specified angle is the angle between the chamfer and the second line of the corner. The specified length is the distance between the "*corner and the intersection of the chamfer*" and the "*second line of the corner.*" (The *second line of the corner* refers to the second line chosen when creating the chamfer.)

Using the Angular Chamfer Tool

1. Select the **Angular Chamfer** tool. The Message Line reads: *Angular Chamfer: Pick first entity [Ctrl = No trim* (Windows), *Option = No trim* (Macintosh)*].*

2. In the Status Line, enter the length for the intersection of the chamfer and the *second line of the corner* from the corner. The default length is .50 inch.
3. In the Status Line also enter the angle you want between the chamfer and the second side. The default Angle is 45°.
4. Click the first curve. The Message Line reads: *Angular Chamfer: Pick second entity. [Ctrl=No trim* (Windows) or *Option=No Trim* (Macintosh)*].*

You can also hold down the SHIFT key and click once inside the corner you want to chamfer. (See the Message Line at the top of your drawing area for a reference to this feature.)

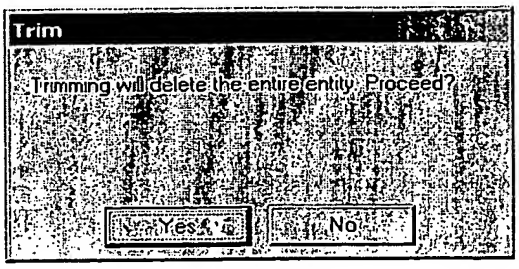
The lines are automatically trimmed or extended by default. If you hold down the CTRL (Windows) or the OPTION (Macintosh) key while you select the objects to chamfer, the objects are not trimmed. (See the Message Line).

The Status Line allows you to set the distance (Length) from the intersection as well as the Angle.

Trim Tools

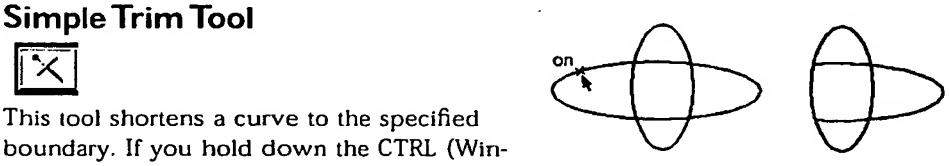
The **Trim** tools on the tool palette lengthen or shorten lines, curves and polylines. You select the limiting object(s) before or after you select the tool. If you don't select everything you need you can hold down the SHIFT key (the tool becomes a temporary **Selection** tool) and click additional objects after the **Trim** tool is selected (see the Message Line).

When trimming curves using one of these tools the following warning will appear if the operation will delete the entire curve.



Click Yes to proceed with the operation or No to exit the operation.

Simple Trim Tool



This tool shortens a curve to the specified boundary. If you hold down the CTRL (Windows) or the OPTION (Macintosh) key while using this tool it becomes the **Relimit** tool.

When you trim a Through Point B-spline or Bezier spline an editable vector spline results. You can move its control points and change its shape and slope. In some situations trimming a curve results in a spline. This spline is also an editable vector spline.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when trimming with this tool.

Rule: Point to what you want to remove.

Using the Simple Trim Tool

1. Select the **Simple Trim** tool. The Message Line reads: *Simple Trim: Select boundaries for simple trim [Shift = Extend].*

   If you already selected the boundaries, proceed to step 3.

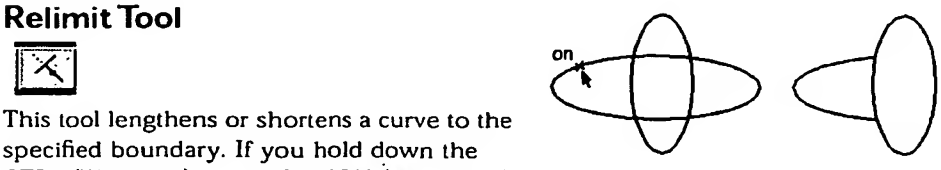
2. Select the object that limits the trim.

   If necessary, use SHIFT-click to select more boundary objects. The Message Line reads: *Simple Trim: Pick section to trim [Shift=Select Boundary, Ctrl=Relimit* (Windows) *Option=Relimit* (Macintosh)*].*

3. Click the section of the object to be discarded.

There are no Status Line entries.

Relimit Tool



This tool lengthens or shortens a curve to the specified boundary. If you hold down the CTRL (Windows) or the OPTION (Macintosh)

key while using this tool it becomes the **Trim** tool. The **Relimit** tool will not relimit arcs.

When you relimit a Through Point B-spline or Bezier spline an editable vector spline results. You can move its control points and change its shape and slope. In some situations relimiting a curve results in a spline. This spline is also an editable vector spline.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when relimiting geometry with this tool.

Rule: Point to what you want to keep.

Using the Relimit Tool

1. Select the **Relimit** tool. The Message Line reads: *Relimit: Select boundaries for relimit [Shift = Extend].*

   If you already selected the boundaries, proceed to step 3.

2. Select the object that limits the change.

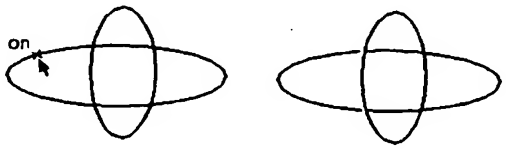
   If necessary, use SHIFT-click to select more boundary objects. The Message Line reads: *Relimit: Pick section to keep [Shift=Select Boundary, Ctrl=Trim* (Windows) *Option=Trim* (Macintosh)*].*

3. Click the section of the object that you want to retain.

There are no Status Line entries.

Segment Tool

This tool divides a curve at intersections with other lines or curves.



When you segment a Through Point B-spline or Bezier spline an editable vector spline results. You can move its control points and change its shape and slope. In some situations segmenting a curve results in a spline. This spline is also an editable vector spline.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when segmenting curves with this tool.

Using the Segment Tool

1. Select the **Segment** tool. The Message Line reads: *Segment: Select boundaries for segment [Shift = Extend].*

   If you already selected the boundaries, proceed to step 3.

2. Select the objects that limits the segmentation.

   If necessary, use SHIFT-click to select more boundary objects. The Message Line reads: *Segment: Pick Entity [Shift=Select Boundary, Ctrl=Current Pen* (Windows) or *Option=Current Pen* (Macintosh)*].*

3. Click the object to be segmented.

The selected object is segmented at the boundary objects. Even though you cannot see the segmentation on the screen, you can select parts of the segmented line by choosing it with the **Selection** tool. In order to see the segmentation on the screen, first select the boundaries and the object you want to segment, then choose ***Edit>Show Points***. The endpoints of the segmented parts are displayed.

If you hold down the CTRL (Windows) or OPTION (Macintosh) key while selecting the line to be segmented the new segment appears in the characteristics of the current pen style (see the Message Line).

There are no Status Line entries.

Divide Tool

This tool subdivides a curve into a specified number of equal-length segments. The left graphic above shows a single line. The right graphic shows the line divided into four segments with the control points displayed.

When you divide a Through Point B-spline or Bezier spline an editable vector spline results. You can move its control points and change its shape and slope. In some situations dividing a curve results in a spline. This spline is also an editable vector spline.

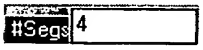
Using the Divide Tool

1. Select the **Divide** tool. The Message Line reads: *Divide: Pick curve to divide [Shift=Extend].*

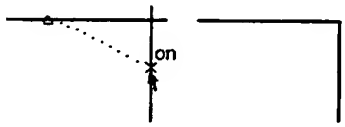
2. Set the #Segs (number of segments) value in the Status Line.
3. Select the object to divide.

   The object divides. The individual segments is not indicated unless you turn on their control points.

The Status Line allows you to set the #Segs in the curve.



Corner Trim Tool



This tool creates a corner from two objects that you specify. Lines are extended or shortened to create the corner.

When you conduct a corner trim on a Through Point B-spline or Bezier spline an editable vector spline results. You can move its control points and change its shape and slope. In some situations trimming a curve results in a spline. This spline is also an editable vector spline.

Using the Corner Trim Tool

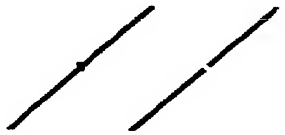
1. Select the **Corner Trim** tool. The Message Line reads: *Corner Trim: Pick portion of first entity to retain. [Shift=Corner, Ctrl=No trim* (Windows) or *Option=No trim* (Macintosh)*].*

2. Select the first curve. The Message Line reads: *Corner Trim: Pick portion of second entity to retain. [Ctrl=No trim* (Windows) or *Option=No trim* (Macintosh)*].*
3. Select the second object.

As an alternative to picking two entities, you can press and hold the SHIFT key and click inside the about-to-be-created corner.

There are no Status Line entries.

Divide at Location Tool



This tool breaks one curve into two at a point that you choose.

When you divide a Through Point B-spline or Bezier spline using this tool an editable vector spline results. You can move its control points and change its shape and slope. In some situations dividing a curve results in a spline. This spline is also an editable vector spline. You can also divide 360° circles and ellipses.

Using the Divide at Location Tool

1. Select the **Divide at Location** tool. The Message Line reads: *Divide at Location: Pick curve to divide at location: [Shift=Extend].*

2. Select the curve. The Message Line reads: *Divide at Location: Pick divide point.*
3. Choose the point on the curve where you want it divided.

There are no Status Line entries.

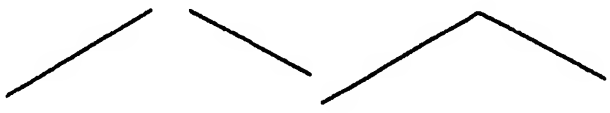
Connect Curve Tool

This tool allows you to connect curves. This Designer Elements program provides you with two options, *Move Curve Point* and *Join*. The *Join* option also includes the ability to set a tolerance. (The tolerance field does not apply to the default *Move Curve Point* option.)

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when connecting curves with this tool.

Using the Connect Curve Tool - Move Curve Point Default

Move Curve Point is the default option when using the tool. Using this option moves the endpoint from



the second selected curve to the endpoint of the first selected curve. This option is useful for creating closed profiles that you can extrude, revolve, or sweep to create solid models.

1. Select the **Connect Curve** tool. The Message Line reads: *Connect Curve: Select two curves to connect end points [Ctrl=Join* (Windows) or *Option=Join* (Macintosh), Shift=Extend*]*.

2. Select the first curve near the endpoint where you want to connect the curve.
3. Select the second curve near the endpoint that you want to move.

   The second curve's endpoint moves to the endpoint of the first curve. The other endpoint of the second curve remains fixed.

The Status Line contains a Tolerance data field. This field only applies to the Join option.

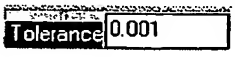
Using the Connect Curve Tool - Join Option

The *Join Curve* option replaces the two curves you select with a new spline curve. The selected lines must share a tangent. This tool will also join two polylines within a specified tolerance distance. This is helpful when you've converted curves into polylines using the *Change Object* command and later want to join them. Select the *Join Curve* option as you pick the two curves.

1. Select the **Connect Curve** tool. The Message Line reads: *Connect Curve: Select two curves to connect end points [Ctrl=Join* (Windows) or *Option=Join* (Macintosh), *Shift=Extend]*.
2. If desired, specify a tolerance distance in the Status Line to which the resulting curves will fit.
3. Hold down the CTRL key (Windows) or the OPTION key (Macintosh) and select the two curves.

   Any discontinuities between the selected lines are eliminated.

The Status Line contains the Tolerance data field.

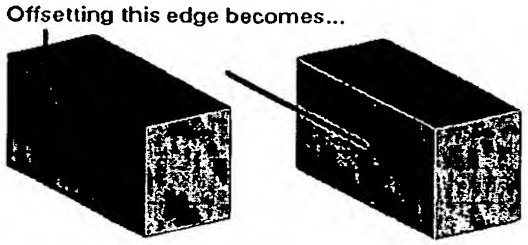


Curve Utility Tools

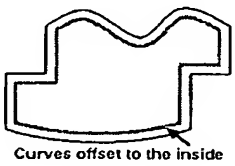
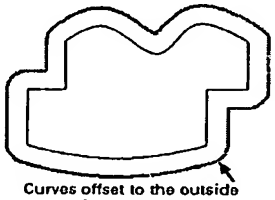
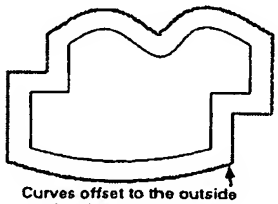
The **Curve Utility** tool palette contains tools for offsetting and exploding curves.

Offset Curve Tool

The **Offset Curve** tool offsets curves by a location or a distance that you specify. The curves can be part of a wireframe object, surface edge (**Offset to Value** tool only) or solid edge. Groups cannot be offset. The graphic here is an example of an offset solid ege.



You can offset one or more curves and supports the following options.

Automatic trimming for curves offset inside-curves.	*Curves offset outside of curves with the arc corners for the offset.*	*Curves offset outside of curves with extended corners for the offset.*
 Curves offset to the inside	 Curves offset to the outside using arc option.	 Curves offset to the outside using the corner option.

The **Offset Curve** tool includes a subpalette with two tools, **Offset to Value** and **Offset by Point**.

When you offset a Through Point B-spline or Bezier spline an editable vector spline results. You can move its control points and change its shape and slope. In some situations offsetting a curve results in a spline. This spline is also an editable vector spline.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when using this tool.

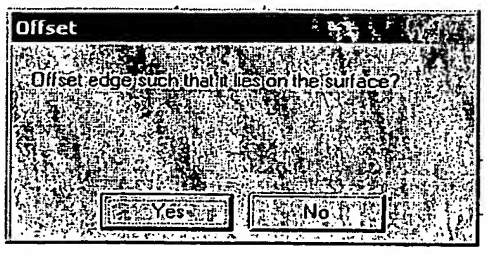
Offset to Value Tool

This tool creates an offset curve at a specified distance from the original curve, surface edge or solid edge. When offsetting surface edges you also have the option of placing the offset on the surface. You can offset one curve or edge or connected curves or edges.

Using the Offset to Value Tool - One Curve, Surface Edge or Solid Edge

1. Select the **Offset Curve** tool.
2. Select the **Offset to Value** tool in the Message Line. The Message Line reads: *Offset Curve: Pick curves to offset distance [Ctrl=Extend Corner* (Windows) or *Option=Extend Corner* (Macintosh), *Shift=Extend].*
3. Enter an offset distance in the Offset data field of the Status Line.
4. Select the curve or edge to offset. The Message Line reads: *Offset Curve: Enter offset direction.*

   If you select a surface edge, the Offset dialog box appears asking if you want the offset to lie on the surface.

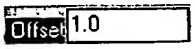
   

   Click Yes to place the offset on surface. Click No and proceed with the next step.
5. Pick a point on either side of the curve or edge to specify the offset

direction. The offset direction must be parallel to the curve or edge. Our Designer Elements program offsets the curve by the distance set in the data field.

While the offset curve is selected you may enter a new offset distance in the Status Line. Press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the Offset distance.



Using the Offset to Value Tool - Connect Curves or Solid Edges

For connected curves or edges you have multiple offset options: extending exterior offsets so they connect with an arc intersection, extending exterior offsets with a corner intersection or trimming intersecting offsets for interior offsets.

1. Select the **Offset Curve** tool.
2. Select the **Offset to Value** tool in the Message Line. The Message Line reads: *Offset Curve: Pick curves to offset distance [Ctrl=Extend Corner* (Windows) or *Option=Extend Corner* (Macintosh), *Shift=Extend].*
3. Enter an offset distance in the Offset data field of the Status Line.
4. Select the curves or edges to offset. The Message Line reads: *Offset Curve: Enter offset direction.*
5. Select the type of offset you desire.

   Exterior Offset - Arc intersection: Pick a point on the outside of the curves or edges.

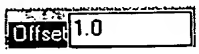
   Exterior Offset - Corner intersection: Hold down the CTRL key (Windows) or the OPTION key (Macintosh) and pick a point on the outside of the curves or edges.

   Interior Offset: Pick a point on the inside of the curves or edges.

   Our Designer Elements program offsets the curves or edges by the distance set in the Status Line data field.

While the offset curves are selected you may enter a new offset distance in the Status Line. Press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the Offset distance.



Offset by Point Tool

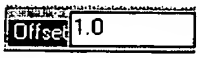
This tool creates an offset at the point specified. You can offset one curve or solid edge or connected curves or solid edges.

Using the Offset by Point Tool - One Curve or Solid Edge

1. Select the **Offset Curve** tool.
2. Select the **Offset by Point** tool in the Message Line. The Message Line reads: *Offset Curve: Pick curves to point [Ctrl=Extend Arc* (Windows) or *Option=Extend Arc* (Macintosh), *Shift=Extend].*
3. Select the curve or edge to offset. The Message Line reads: *Offset Curve: Enter offset point.*
4. Pick the desired point for the offset. Our Designer Elements program offsets the curve to that point. The offset value is displayed in the Offset data field in the Status Line.

While the offset curve is selected you may enter a new offset distance in the Status Line. Press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the offset distance.



Using the Offset by Point Tool - Connected Curves or Solid Edges

1. Select the **Offset Curve** tool.
2. Select the **Offset by Point** tool in the Message Line. The Message Line reads: *Offset Curve: Pick curves to point [Ctrl=Extend Arc* (Windows) or *Option=Extend Arc* (Macintosh), *Shift=Extend].*
3. Select the curves to offset. The Message Line reads: *Offset Curve: Enter offset point.*
4. Select the type of offset you desire.

   Exterior Offset - Arc intersection: Pick a point on the outside of the curves.

   Exterior Offset - Corner intersection: Hold down the CTRL key (Windows) or the OPTION key (Macintosh) and pick a point on the outside of the curves.

   Interior Offset: Pick a point on the inside of the curves.

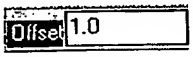
   Our Designer Elements program offsets the curves at that point. The offset value

is displayed in the Offset data field in the Status Line.

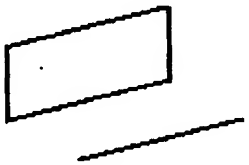
5. Pick the desired point for the offset. Our Designer Elements program offsets the curve to that point. The offset value is displayed in the Offset data field in the Status Line.

While the offset curves are selected you may enter a new offset distance in the Status Line. Press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the offset distance.



Extrude Curve Tool



This tool copies a curve at a distance and in the direction that you specify. It connects its endpoints by two lines back to the original curve. In the graphic to the right the lower curve is extruded to form the polygon.

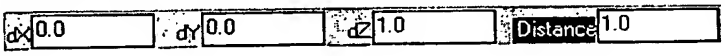
Using the Extrude Curve Tool

1. Click the **Extrude Curve** tool. The Message Line reads: *Extrude Curve: Pick curves to extrude [Shift=Extend].*

2. Select the curve(s) to extrude. The Message Line reads: *Extrude Curve: Specify two points for extrusion direction.*
3. Click two points to indicate where the new extruded curve will be placed. The direction of the extrusion is specified by the order in which you pick the points. This Designer Elements program applies the distance and direction you specify.

While the object is selected you can change the extrusion distance in the Status Line. Press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the extrusion's dX, dY and dZ values and the Distance.



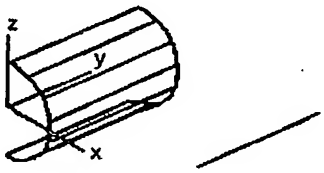
Tip:

To extrude an object you must click to define the extrusion rather than drag.

To extrude a curve at an angle not normal to the curve, change the view. Example: Create a rectangle in the top plane. Change the plane to the front plane and place the extrude points to define an angle and length.

Revolve Curve Tool

This tool revolves a curve around an axis.



Using the Revolve Curve Tool

1. Select the **Revolve Curve** tool. The Message Line reads: *Revolve Curve: Pick curves to revolve [Shift=Extend].*

2. Select the curve(s) to revolve. The Message Line reads: *Revolve Curve: Specify revolution origin.*
3. Click a point about which to revolve the curve. The Message Line reads: *Revolve Curve: Specify revolution axis.*
4. Click two points to indicate the axis of revolution. The direction of the revolution is specified by the order you pick the points. This Designer Elements program rotates the curve about the axis by the number of steps and through the angle set in the data fields on the status line.

While the object is still selected you can change the number of steps and the angle of revolution in the Status Line. Press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the number of steps and the angle of revolution.

Tech Note:

You cannot revolve a *Smart Polygon*. It must first be converted into single lines using the Change Object Type command. See Chapter 12 for more information on polygon types.

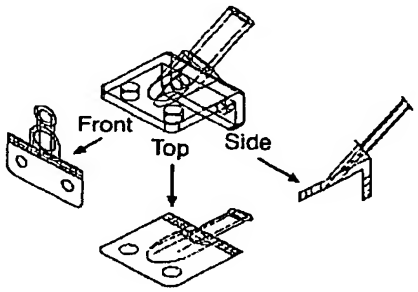
To define the axis of revolution you must click the points rather than drag.

Project Curve Tool

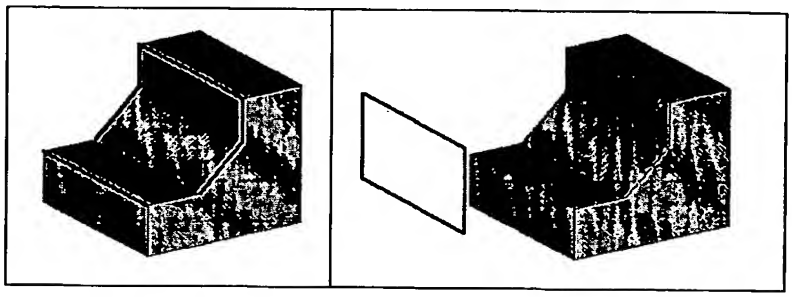
The **Project Curve** tool projects 3D curves, including solid edges onto a 2D plane. Arcs, circles and ellipses maintain their precise shape when you project them normal to their definition. If you project them otherwise a spline is used with a default curve tolerance of 0.001 inches. This spline is an editable vector spline.

When you project a Through Point B-spline or Bezier spline an editable vector spline results. You can move its control points and change its shape and slope. In some situations projecting a curve results in a spline.

The select curves are created on the plane and replace the original curves the curves unless the CTRL key (Windows) or OPTION key (Macintosh) is held down. The graphic here shows a model projected to XY, XZ and YZ planes.

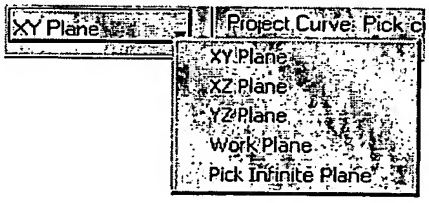


The left graphic below shows a model with four solid edges selected (displayed in black). The right graphic shows the edges projected to the XZ Plane.

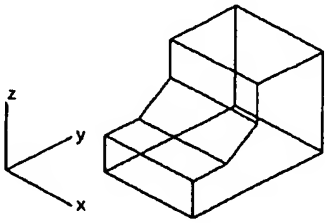


Plane Options

The pull-down menu in the Message Line provides five options for choosing the plane: *XY Plane*, *XZ Plane*, *YZ Plane*, *Work Plane* and *Pick Infinite Plane*.

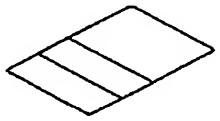


The following figures, displayed in the Isometric view, explain the projection options. All graphics used for the individual planes are also displayed in the Isometric view.



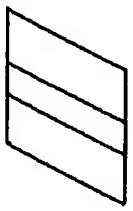
XY Plane

Choosing this option projects the selected curves into the XY plane (Top) where X=0 and Y=0.



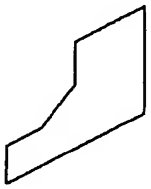
XZ Plane

Choosing this option projects the selected curves into the XZ plane (Front) where X=0 and Z=0.



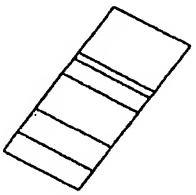
YZ Plane

Choosing this option projects the selected curves into the YZ plane (Side) where Y=0 and Z=0.



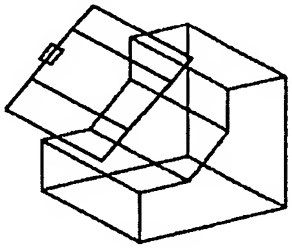
Work Plane

Choosing this option projects the selected curves into the work plane. For this example, the work plane was defined using the sloped face of the object.



Pick Infinite Plane

Choosing this option project the selected curves to the specified infinite plane.



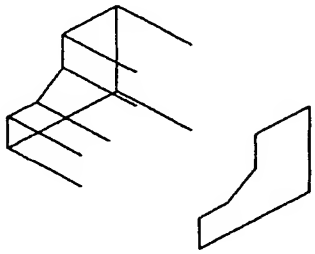
Use the Plane commands in the Plane menu to define your own plane. You must set the work plane to the desired plane before using this option.

Projection Options

When you project curves this Designer Elements program takes those curves and creates new geometry on the specified plane. You have two options, Default and Copy (see the Message Line).

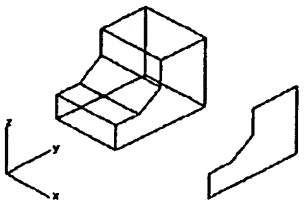
Project Curves - Default Option

This option takes the selected geometry. See the graphic here. Notice that the selected geometry no longer appears on the original part.



Project Curves - Copy Option

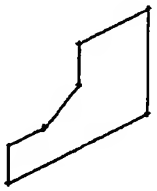
This option copies the selected geometry. To choose this option hold down the CTRL key (Windows) or OPTION (Macintosh). See the graphic here.



Projecting & Multiple Curve Selection

After you select the tool you can select multiple lines for projection by holding down the SHIFT key and selecting the desired lines, as noted in the Message Line.

You can also select multiple lines or an entire object by dragging a selection fence around the entire object. When you project the object all curves that are normal to the projection plane are recreated as points in the new curve. The graphic here shows a YZ projection. The cross hairs represent the points.



Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when projecting curves with this tool.

Using the Project Curve Tool

1. Select the **Project Curve** tool. The Message Line reads: *Project Curve: Pick curves to project [Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh), *Shift=Extend].*

2. From the pull-down menu in the Status Line choose the projection plane.
3. Select the curves to project. Hold down the SHIFT key or drag a selection fence to select multiple lines. The Message Line now reads, *Project Curve: Specify point in projection plane [Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*].*
4. Pick the point on the projection plane where the curves should be projected. The selected lines are replaced with lines projected on the selected plane.

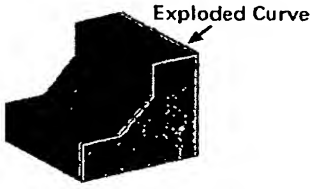
   To project a copy of the lines hold down the CTRL key (Windows) or OPTION key (Macintosh) before selecting the point.

Tech Note:

If you hit ENTER (Windows) or RETURN (Macintosh) instead of specifying a point, the projection is placed at the origin.

There are no Status Line entries.

Explode Edge Tool



This tool enables you to create a curve from a surface or solid edge. For surfaces and solids, you can also explode Iso lines when displayed on your geometry. This ability is useful when constructing geometry off an object.

When you explode a solid edge, solid Iso line, surface edge or surface Iso line the new curve is placed on the work layer and assumes its color characteristic. If the layer has no specific color characteristic, the curve displays in the color selected in the Pen menu. If the exploded edge or Iso line results in a spline, a vector spline results. You can move its control points and change its shape and slope.

Using the Explode Edge Tool

1. Select the **Explode Edge** tool. The Message Line reads: *Explode Edge: Pick edges to explode [Shift=Extend].*

2. Select the surface or solid edge or Iso line from which you want to create a curve.

   The curve is created at the same location as the selected edge.

There are no entries in the Status Line.